

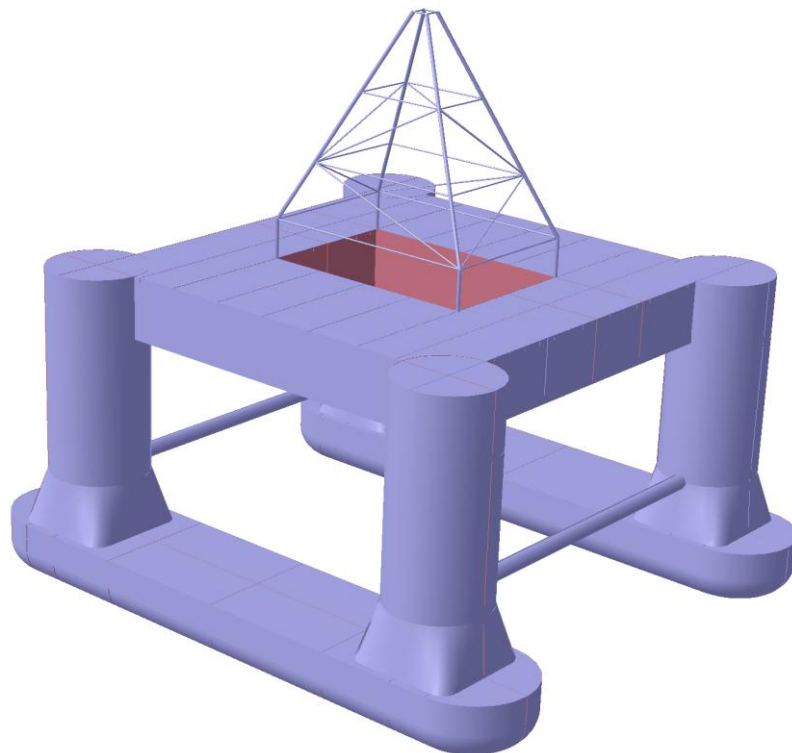


SESAMTM

USER MANUAL

GENIE VOL. III

MODELLING OF PLATE/SHELL STRUCTURES



Concept design and analysis
of offshore & maritime structures

DET NORSKE VERITAS

SesamTM

User Manual

GeniE Vol. III

Modelling of plate & shell structures

Concept design and analysis
of offshore & maritime structures

15 September 2011

Valid from program version 6.0

Developed and Marketed by
DET NORSKE VERITAS

DNV Software Report No.: 00-000 / Revision 0, 15 September 2011

Copyright © 2011 Det Norske Veritas Software

All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the publisher.

Published by:

Det Norske Veritas Software
Veritasveien 1
N-1322 HØVIK
Norway

Telephone: +47 67 57 99 00

Facsimile: +47 67 57 72 72

E-mail, sales: software.sesam@dnv.com

E-mail, support: software.support@dnv.com

Website: www.dnvsoftware.com

If any person suffers loss or damage which is proved to have been caused by any negligent act or omission of Det Norske Veritas, then Det Norske Veritas shall pay compensation to such person for his proved direct loss or damage. However, the compensation shall not exceed an amount equal to ten times the fee charged for the service in question, provided that the maximum compensation shall never exceed USD 2 millions. In this provision "Det Norske Veritas" shall mean the Foundation Det Norske Veritas as well as all its subsidiaries, directors, officers, employees, agents and any other acting on behalf of Det Norske Veritas.

Modelling of plate & shell structures

Table of Contents

1. INTRODUCTION.....	5
1.1 HOW TO READ THIS MANUAL.....	5
1.2 LEARNING FROM TUTORIALS FOR CODE CHECKING	6
1.3 ACRONYMS FREQUENTLY USED IN THE MANUAL	8
2. CONCEPT MODELLING STRATEGIES.....	9
2.1 MAKING DIFFERENT ANALYSIS REPRESENTATIONS	9
2.2 STRUCTURE MODELLING STRATEGIES	11
2.3 LOAD APPLICATION STRATEGIES.....	12
2.4 MESHING STRATEGIES	14
3. PLATE & SHELL STRUCTURES	17
3.1 THE DESIGN PREMISE	17
3.1.1 <i>Material properties</i>	17
3.1.2 <i>Section properties</i>	19
3.1.3 <i>Thickness properties</i>	19
3.2 CREATE GUIDING GEOMETRY	20
3.2.1 <i>Guide plane</i>	20
3.2.2 <i>Guide point</i>	21
3.2.3 <i>Guide line</i>	22
3.2.4 <i>Guide Spline</i>	23
3.2.5 <i>Poly-curve</i>	23
3.2.6 <i>Guide arc elliptic</i>	25
3.2.7 <i>Guide circle</i>	25
3.2.8 <i>Model curve</i>	26
3.2.9 <i>Fillet curve</i>	27
3.2.10 <i>Model guiding geometry using existing snap features</i>	29
3.2.11 <i>Find, select and display guiding geometry</i>	30
3.2.12 <i>Delete, Move and copy guiding geometry</i>	33
3.2.13 <i>Modify guiding geometry</i>	43
3.2.14 <i>Snap points</i>	47
3.3 CREATE A STRUCTURE CONCEPT MODEL	48
3.3.1 <i>Basic modelling features</i>	48
3.3.1.1 <i>Insert using manual input</i>	49
3.3.1.2 <i>Insert using skin</i>	50
3.3.1.3 <i>Insert using flat region</i>	52
3.3.1.4 <i>Insert using loft</i>	57
3.3.1.5 <i>Insert using cover curves</i>	61
3.3.1.6 <i>Insert using extrude</i>	63
3.3.1.7 <i>Insert using curve-net interpolation</i>	65
3.3.1.8 <i>Define plates using beams as reference</i>	69
3.3.1.9 <i>Divide using existing structure</i>	70
3.3.1.10 <i>Divide and trim using existing structure</i>	71
3.3.1.11 <i>Divide using guide curves</i>	72
3.3.1.12 <i>Punch using guide curves</i>	72
3.3.1.13 <i>Punch and divide using a profile</i>	73
3.3.1.14 <i>Divide using planes</i>	74
3.3.1.15 <i>Join plates</i>	75
3.3.2 <i>Use 2D structure to make 3D structures</i>	77
3.3.3 <i>Make a 3D tubular joint</i>	81
3.3.4 <i>Make special 3D structures like cones, spheres and bulbs</i>	82
3.3.5 <i>Add stiffeners to plates and shells</i>	84
3.3.5.1 <i>Straight beams between snap points</i>	84
3.3.5.2 <i>Curved beams in-between snap points</i>	84
3.3.5.3 <i>Straight or curved beams from guide curves</i>	84
3.3.5.4 <i>Straight or curved stiffeners from plate or shell edges</i>	85

3.3.5.5	Orientation of beams and stiffeners	85
3.3.5.6	Flush stiffeners	87
3.3.6	Topology, edges and vertices	88
3.3.7	Verify the concept model	90
3.3.8	Stand-Alone Beams	93
4.	MASSES, LOADS AND COMPARTMENTS.....	94
4.1	LOAD CASES AND LOAD COMBINATIONS	95
4.2	EXPLICIT LOADS ON BEAMS AND STIFFENERS	97
4.2.1	Point loads on beams	97
4.2.2	Point loads on plates	98
4.2.3	Line loads on beams	99
4.2.3.1	The line load as a separate object	100
4.2.3.2	The line load referencing the beam or stiffener	101
4.2.3.3	Generic line loads	102
4.2.4	Line loads on plate edges	104
4.2.5	Temperature loads	106
4.2.6	Prescribed displacements	107
4.3	SURFACE LOADS	108
4.3.1	Plate surface loads	108
4.3.1.1	Pressure loads	109
4.3.1.2	Traction loads	111
4.3.1.3	Component load	112
4.3.1.4	Modify and verify surface loads	113
4.3.2	Surface loads on shells	114
4.3.3	Define wet surfaces	114
4.3.3.1	Pressure loads	115
4.3.3.2	Traction loads	119
4.3.3.3	Component loads	120
4.3.3.4	Transfer hydro pressure data to HydroD	121
4.4	COMPARTMENT LOADS	123
4.4.1	Create compartments	124
4.4.1.1	Visualise and rename	124
4.4.1.2	Modify structure	126
4.4.1.3	Open compartments and non-structural plates	127
4.4.1.4	Non-watertight bulkheads	128
4.4.1.5	Corrosion addition	129
4.4.2	Design load based analysis – manual load application	130
4.4.2.1	Content	130
4.4.2.2	Liquid content	130
4.4.2.3	Solid content	133
4.4.2.4	Manually defined compartment loads	135
4.4.3	Design load based analysis – rule based load application	136
4.4.4	Direct analysis - transfer compartment data to HydroD	138
4.5	EQUIPMENT LOADS	140
4.5.1	Create equipments	140
4.5.2	Editing the COG and the footprint	140
4.5.3	Placing the equipment	142
4.5.4	Creating forces from placed equipments	143
4.5.5	Verify the applied loads	147
4.6	MASSES AND INERTIA LOADS	148
4.6.1	Structural mass	148
4.6.2	Point mass	150
4.6.3	Equipment mass	151
4.6.3.1	Mass model for hydrodynamics	151
4.6.3.2	Mass model for structural dynamics	153
4.6.3.3	Mass model when neglecting eccentricities	154
4.7	VERIFY AND DOCUMENT LOADS AND MASSES	155
5.	APPLY BOUNDARY CONDITIONS.....	159
5.1	FIXATION AND ROTATION SUPPORTS	160
5.2	GROUND SPRINGS	162
5.3	BOUNDARY STIFFNESS MATRIX	162

5.4	CREATE A SUPER-ELEMENT	163
5.5	LOCAL CO-ORDINATE SYSTEMS	163
5.6	RIGID LINK SUPPORT	165
5.6.1	<i>Rigid body behaviour</i>	166
5.6.2	<i>User defined behaviour</i>	168
6.	MAKE AND CONTROL THE FINITE ELEMENT MESH	169
6.1	GENERAL	169
6.1.1	<i>Refine mesh by inserting a beam</i>	170
6.1.2	<i>Refine mesh by inserting a plate</i>	170
6.1.3	<i>Refine mesh by inserting feature edges</i>	171
6.1.4	<i>Labelling</i>	171
6.1.5	<i>Documenting the finite element mesh</i>	172
6.1.6	<i>Colour coding</i>	173
6.1.7	<i>Element types</i>	174
6.2	GLOBAL MESH SETTINGS	175
6.2.1	<i>Mesh settings - general</i>	176
6.2.1.1	<i>General FEM options</i>	176
6.2.1.2	<i>Model topology</i>	177
6.2.1.3	<i>Element preferences</i>	178
6.2.1.4	<i>Idealisations</i>	179
6.2.1.5	<i>Corrosion addition</i>	181
6.2.1.6	<i>Corrosion addition – CSR Bulk example</i>	183
6.2.1.7	<i>Other preferences</i>	184
6.2.1.8	<i>Naming preferences</i>	185
6.2.1.9	<i>Face mesher</i>	185
6.2.1.10	<i>Edge mesher</i>	187
6.2.1.11	<i>Scantling idealizations</i>	188
6.2.1.12	<i>Idealisations</i>	189
6.2.2	<i>Mesh settings – max/min angles</i>	190
6.2.3	<i>Mesh settings – Jacobi</i>	190
6.2.4	<i>Mesh settings – eliminate edge</i>	192
6.2.5	<i>Mesh settings – chord height</i>	193
6.3	LOCAL MESH SETTINGS	195
6.3.1	<i>Number of elements along a line</i>	195
6.3.2	<i>Feature edges</i>	196
6.3.3	<i>Mesh options for face or edge</i>	197
6.4	REFINE MESH ZONES	200
6.5	PRIORITIZED MESHING	204
6.6	MESH PARTS OF STRUCTURE ONLY	208
6.7	MESH LOCKING	210
7.	RUN ANALYSIS	212
7.1	PRE-DEFINED ANALYSIS ACTIVITIES	212
7.2	EDIT ANALYSIS ACTIVITIES	215
7.3	STATIC, EIGENVALUE AND DYNAMIC ANALYSIS	217
8.	RESULTS PROCESSING	220
9.	IMPORT AND EXPORT	223
9.1	THE COMMAND FILE GENERATED BY GENiE	224
9.2	THE CONDENSED COMMAND FILE	224
9.3	THE XML CONCEPT MODEL FILE	225
9.4	FROM/TO FEM	226
9.5	IMPORT SACS FILE	229
9.6	THE ACIS SAT FILE	232
9.7	THE GENSOD FILE	233
10.	PICTURES AND REPORTS	234
11.	APPENDIX A: REFERENCES	236
12.	APPENDIX B: MATERIAL LIBRARY	237

1. INTRODUCTION

This is the user manual for the part of GeniE dedicated to modelling of plate and shell structures for offshore and maritime structures.

This user manual assumes that the user has knowledge in the use of GeniE as covered by the GeniE User Manual Volume I (the main user manual).

This manual describes how to create plate and shell structures based on guiding geometry or existing information, how to apply loads and boundary conditions, how to create and control the finite element, how to run analysis as well as how to look at results.

Chapter 2 gives an overview of concept modelling strategies while the rest of the Chapters focuses load applications, boundary conditions, how to create and control a finite element mesh, how to run analysis and finally how to look at results.

1.1 How to read this manual

Chapter 2 *Concept modelling strategies* gives an overview on the importance of deciding the concept modelling strategies for plate and shell structures. This Chapter should be read carefully to get an overview on alternatives and also various strategies that can be selected before or during the modelling.

The rest of the Chapters give relevant examples on the functionality used to make plate and shell structure. For small frame models the user is advised to follow the steps in the GeniE User Manual Volume I (the main user manual). For analysis involving wave and pile/soil analysis (typically jackets and jack-ups) the user should also consult the GeniE User Manual Volume II.

A command from the menu list (also referred to as the pulldown menu) is written like this:

Insert/Beam/Dialog. The name of a tool button is written like this: **Basic plate**. A function buttons is referred to like this: **F1**.

GeniE comes with a context sensitive menu. You invoke this menu by pushing your right mouse button when the mouse is located above a selected object. In this manual this operation is termed **RMB**. The commands on the context sensitive menu are written like this: *Join Beams*.

Viewing this manual assumes the usage of Adobe Acrobat Reader version 8.0 or higher. You may use older versions, but then you don't have access to important features like e.g. free text search and bookmarks (table of content + hyperlinks).

It is particularly noted that this User Manual documents all capabilities of GeniE. If you do not have access to the program extensions "*Curved structure modelling*", "*Code checking of beams*" and "*Code checking of plates*" there are several items in this manual you do not have access to in your program. These features are blanked out in your program version. To run analysis you need to have Sestra, Wajac and Splice installed depending on the type of analysis.

1.2 Learning from tutorials for code checking

GeniE comes with an on-line help system (**Help/Help Topics** or **F1**). Its purpose is to provide easy access to release notes, limitations, tutorials, wizards and this user manual. In addition it contains a detailed documentation of all available commands in the journalling system (based on J-script). There are also videos showing how to do certain operations, these are best viewed using resolution 1280x1024.

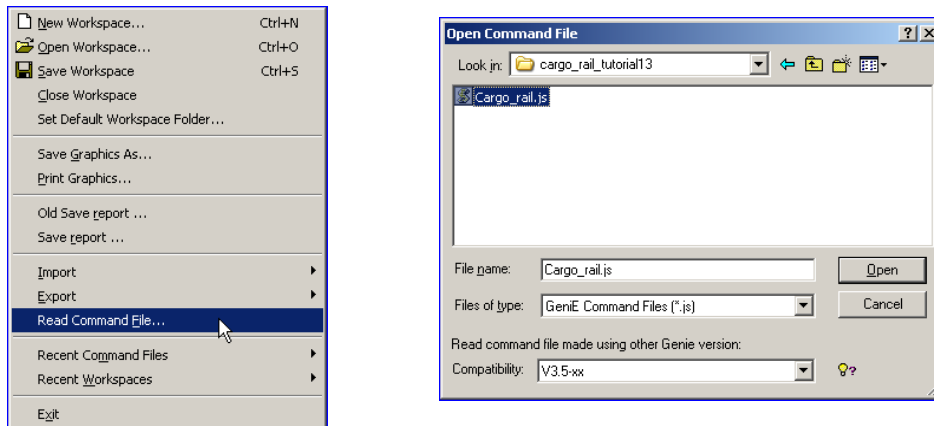
The easiest way to find the tutorials is from GeniE's help page. For modelling of plate and shell structures the most relevant tutorials are found under the category "GeniE Tutorials – Advanced Modelling".

<p>GeniE Version: D4 6-11 19 May 2008 Copyright (c) 1999-2008 DHV Software</p> <p>Introduction Introduction Release Notes Support Request</p> <p>User's Guide Vol 1 - Concept engineering Vol 2 - Waves, pile and soil Vol 3 - Plate/Shell Structures Vol 4 - Beam code checking Vol 5 - Plate code checking Reference Documents</p> <p>Command Reference JScript commands</p> <p>Tutorials Example Index</p> <p>Wizards Wizard templates</p> <p>HowTo- Videos Video Index</p>	<p>GeniE Tutorials - Basic and Codechecking</p> <div>  <p>A small introduction to GeniE - For new users you should do this tutorial first.</p> <p>Genie Basic Workshop</p> </div> <div>  <p>Learn the user interface and how to do a small modelling and analysis task.</p> <p>Genie Workshop Input files</p> </div> <div>  <p>Make a small module frame and load with explicit loads and equipments. Run analysis, perform code checking using Framework as an integrated service.</p> <p>Genie Frame Workshop Framework Workshop Input files</p> </div> <div>  <p>This tutorial will take you through the steps of modelling and analysing an arched steel building frame.</p> <p>Genie Lite Workshop Input files</p> </div> <div>  <p>Make a structure built up with beams and loaded with equipments. Second part of tutorial includes one joint modelled with curved plates.</p> <p>Deck Modelling Input files</p> </div> <div>  <p>Learn how to do code checking of beams in a topside structure. The tutorial is based on APIWSD.</p> </div>	<p>GeniE Tutorials - Advanced Modelling</p> <div>  <p>Make a crane pedestal sitting on top of a vessel. The structure is modelled with curved plates.</p> <p>Crane Pedestal Input files</p> </div> <div>  <p>Make the pontoons and column transitions using curved plates and stiffeners. Focus is also on controlling the finite element mesh.</p> <p>Semisub Pontoon Input files</p> </div> <div>  <p>The purpose of this workshop is to create two models of a tubular joint – one beam model and one 3D shell model – and compare the results to compute stress concentration factors.</p> <p>Tubular Joint Modelling Input files</p> </div> <div>  <p>This tutorial gives one example on how the script language can be used to create parametric models.</p> <p>Semisub Panel Model Input files</p> </div> <div>  <p>Make a cargo rail analysis by modelling the aft part of a typical vessel. Main focus is on modelling, but there is also a loadcase so that analysis can be done.</p> <p>Cargo Rail Input files</p> </div> <div>  <p>Panel Code Check Learn how to perform a buckling check according to CSR Bulk.</p> </div>
---	--	--

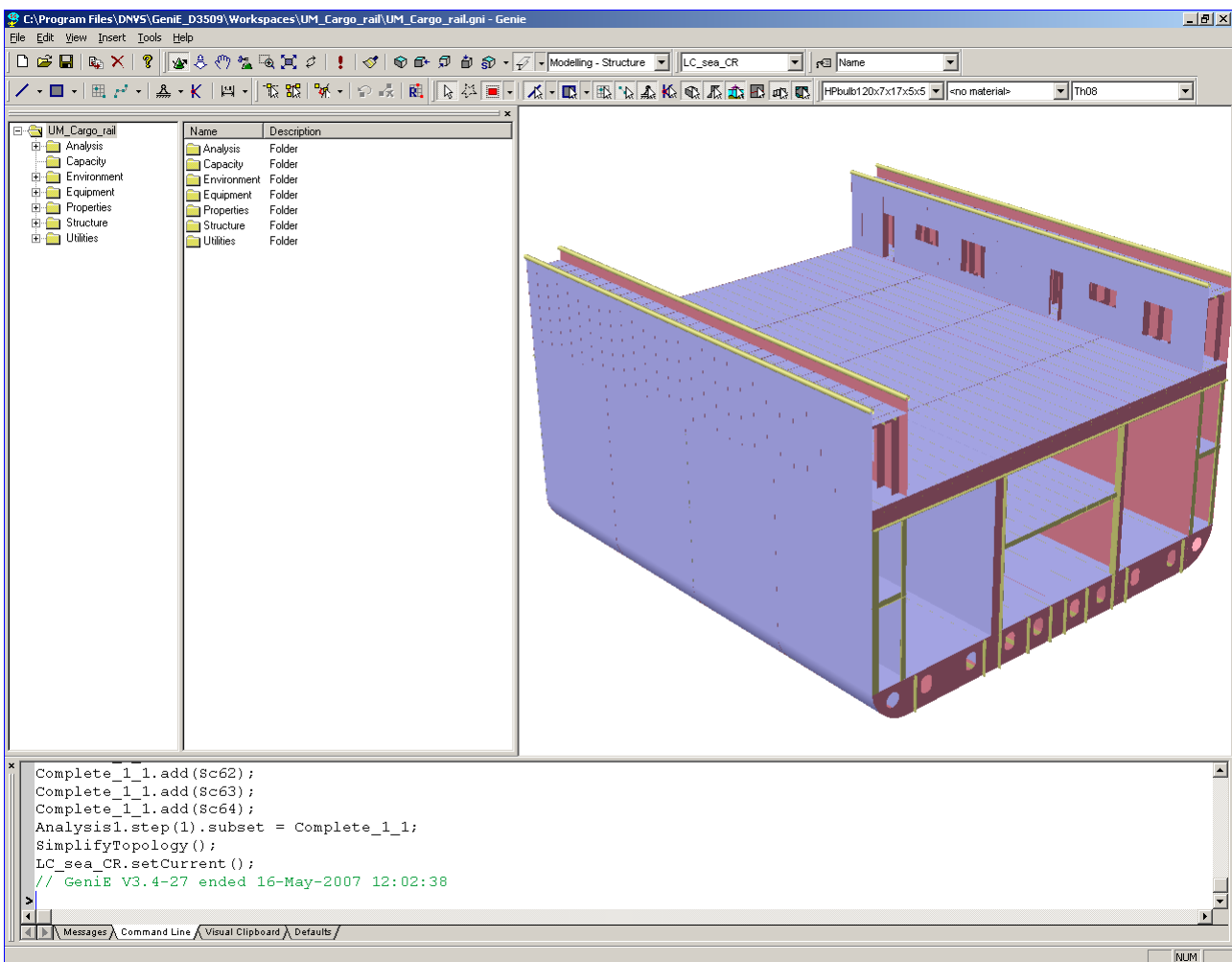
The most efficient way to work with the tutorials is to make a print-out of the tutorials, start GeniE, create a new workspace (command **File/New workspace**) and follow the steps in the tutorial. Each tutorial comes with a pre-defined journal file (command file) – you find these from the help page as shown above under "Input Files". If you want to use these files the steps are as follows:

1. Create a new workspace **File/New Workspace<name>**
(Keep the default settings for tolerant modelling and database units)
2. Read in the journal file from **File/Read Command File|<browse until you find the desired input file>**
3. Save your model by **File/Save**
4. You have now created the same model as in the tutorial you selected.

In the example below the <name> has been set to *UM_Cargo_rail* and the imported file is for the tutorial “*Cargo_rail.js*”.



The sequence above creates the following view in GeniE (the colour background has been set to white and the view is set to “Modelling Structure”):



You may also read in a journal file by using drag-and-drop. Simply select a journal file from your browser and drop it into the command line window.

1.3 Acronyms frequently used in the Manual

Acronym	Explanation
RMB	Right Mouse Button
LMB	Left Mouse Button
GUI	Graphical User Interface
DOF	Degree Of Freedom
CLI	Command Line Interface
LJF	Local Joint Flexibility
FEM file	SESAM Input Interface File
Panel model	A FEM file intended for use in hydrodynamic analysis in HydroD
Plate	Normally a “flat” plate (in one plane)
Shell	Normally a plate with single or double-curvature
Beam	Normally a beam not aligned with plate or shell
Stiffener	Normally a beam connected to a plate or shell
Design load based analysis	All loads are created inside GeniE and a linear analysis can be done. The loads are often defined by rules
Direct analysis	GeniE will create necessary model data to be used in superelement analysis or together with HydroD in a hydrodynamic analysis
CSR	Common Structural Rules
COG	Centre of gravity

2. CONCEPT MODELLING STRATEGIES

This manual describes how to

- make guiding geometry
- create plate and shell structures (including stiffeners) using the guiding geometry
- apply loads to the plates or shells
- make and control a finite element mesh
- add boundary conditions and run analysis

This user manual also described how to import from other systems (CAD) and which are the steps necessary to make a GeniE concept model.

Before you start to make your concept model, you should decide which strategies you should use for modelling, load applications and how to make analysis representations – these are described in the following.

A detailed description on how to do the various tasks is described in the next chapter.

2.1 Making different analysis representations

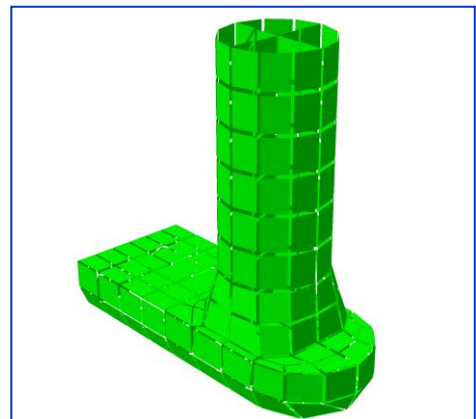
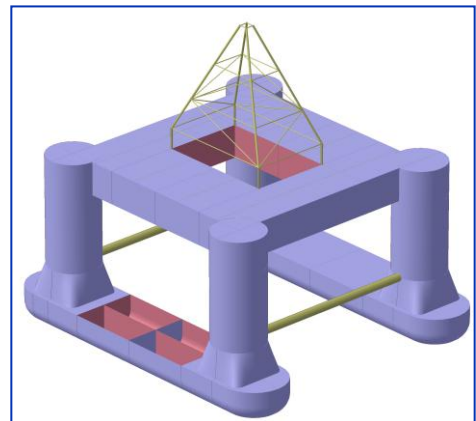
The same concept model created in GeniE may be used for global or local models. This user manual describes how to change meshing parameters to make different analysis representations (FE models) for the same concept model. A concept model can also be used to make one or several super-elements for use in super-element analysis. The principles behind these options are illustrated using a semi-submersible as a case example.

A concept model of a semi-submersible contains regular plates, curved plates, beams and stiffeners. In addition, there are attributes for load generation (typically compartment content or explicit loads like pressure or line loads).

This concept model can be used as basis for panel model (i.e. the model as used by HydroD for stability and wave load analysis), in global and local finite element analyses, as well as in detailed fatigue analysis.

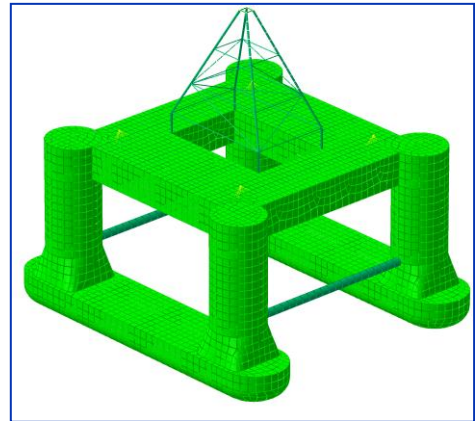
Wave load analysis in HydroD allows for symmetric models. To make a panel model of a sub-set of the concept model there are features for creating analysis models of named sets only. A panel model has normally a coarse mesh density; hence the mesh settings are normally different from a structural analysis model.

The picture to the right shows a panel model which has been created by meshing $\frac{1}{4}$ of the submerged structure. One mesh setting has been used; in this case a characteristic value of 5 m.



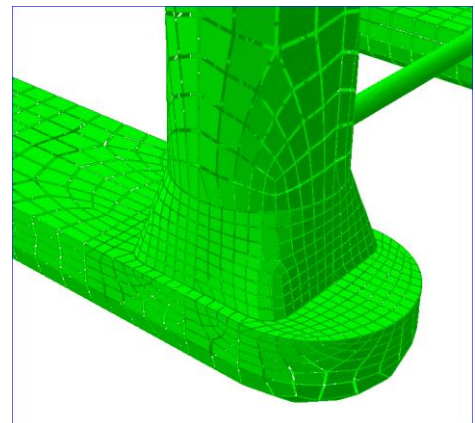
A global structural analysis is normally carried out to find the overall strength capacity of the structure. For such analysis the entire structure is analysed using either a direct load approach (all loads are manually defined by the user) or an integrated load approach where wave load effects (pressure and accelerations) are computed by HydroD and seamlessly used by the analysis tool Sestra. In the latter case, the modelling, wave load generation and analysis should be run from BriX Explorer configured for Sesam using GeniE, HydroD and Sestra.

The global FE model as shown to the right contains all structural parts and has one global mesh default setting (in this case 3 m).



Local analyses are performed to document strength on a more detailed level as a result from the global structural analysis, requirements in rules and regulations or from engineering experience.

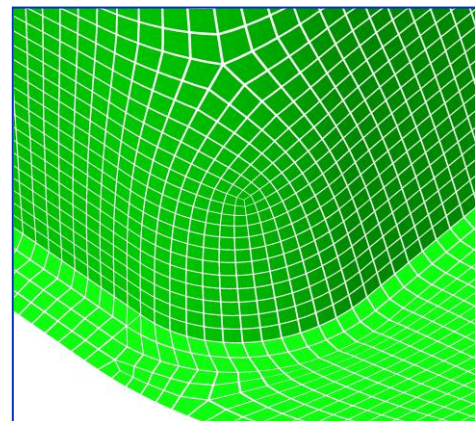
In the example to the right, the area of interest has a more dense mesh setting (1 m) than the rest. Furthermore, the local area has been meshed prior to the other structure to ensure a best possible FE model in this area.



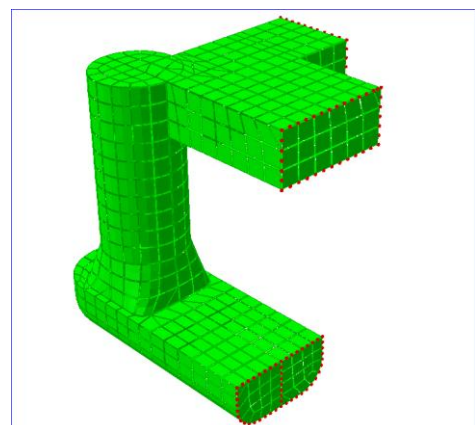
The local model may be analysed together with the rest of the structure or by using sub-modelling techniques in Submod (use BriX Explorer configured for Sesam to control the workflow process).

For fatigue analysis or when computing SCF's (Stress Concentration Factor) a mesh size equal to the plate thickness is often used. In the example to the right, a part of the structure has been meshed with finite element size equal to the plate thickness.

In this case the finite element model has been analysed using a sub-modelling approach where prescribed displacements along the sub-model's boundaries are computed in a global analysis and automatically applied to the local model.



A superelement for use in a superelement assembly is made when setting the right properties for boundary conditions (highlighted in the picture to the right). It is possible to make several superelements from the same concept model by referencing to named sets when making the analysis models.



2.2 Structure modelling strategies

GeniE opens up for a variety of modelling strategies depending on the task you are going to do or how much and which type of information you possess before you start. Typically, the strategies are:

- Start with guiding geometry such as lines or curves and use these to make beams or plates. This modelling approach is often referred to as “bottom-up” modelling.
- If you have a concept model you may divide and split this model to modify or add details to the original model. This technique is often called “top-down” modelling. In most cases the “bottom-up” and “top-down” modelling is used together allowing you to add details or do modification at any stage in your modelling tasks. As such, GeniE is well suited for frequent model changes in the engineering design phase.
- It is often beneficial to re-use existing concept models created in previous GeniE sessions. There are multiple alternatives for importing such data; either from using a journal file (or parts of it) or by importing concept model as found on the XML file.
- You may also use finite element model data from other systems like previous modelling Sesam tools (typically Preframe and Prefem) as well as Sacs and StruCad3D. The finite element models will be converted to concept models. Notice that there are limitations with regard to which data item can be imported.
- There are also features for importing relevant data from CAD systems. The solutions supported are via the sdnf format (beam models created in PDS and PDMS), the SAT format (neutral format made by Spatial) and guidelines as defined on the DXF format. Notice that import of CAD data has limitations on converting types of data items.
- In case you are using Section Scantlings from Nauticus Hull you can automatically create the mid-ship section in GeniE (longitudinal plates and stiffeners) between two frames.

Some of the above modelling techniques are briefly explained in the following.

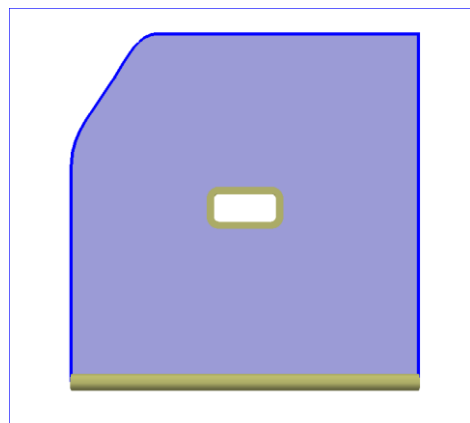
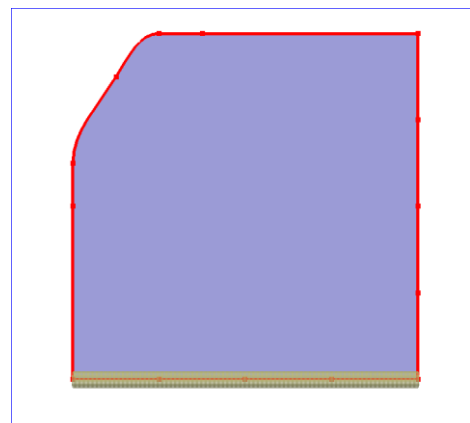
In the picture to the right, three guiding lines have been created to be used in “bottom-up” modelling. The beam has been created by referring to the relevant guide line. The plate has been created by a cover operation using all three guide lines.

In this case, all the lines lie on the same horizontal plane; GeniE supports full 3D modelling of plates and beams/stiffeners like you will find on many offshore and maritime structures.

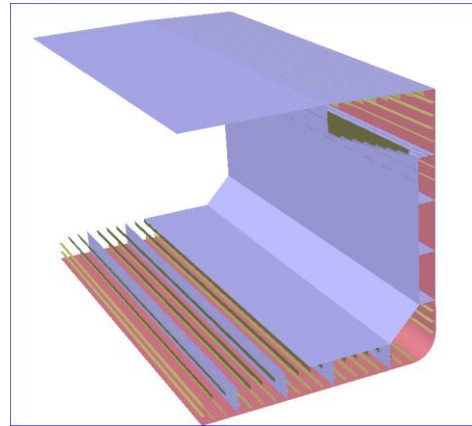
A “top-down” modelling approach has been used to create the plate with the hole as shown to the right. A punching operation has been used to create the hole and its flanges.

Another typical example of “top-down” modelling is to divide the plate to the right and use different plate thicknesses to the various parts.

The “bottom-up” approach may also be used to create similar structure by building up smaller parts and joining them afterwards.

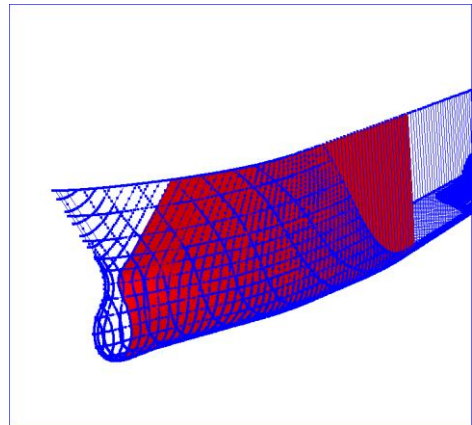


When importing data from Nauticus Hull Section Scantlings, the longitudinal structural parts between the two selected frames (i.e. start and stop) in the mid-ship area are automatically defined. The concept model contains plate thicknesses, stiffener properties, alignments and rotations (i.e. stiffeners are perpendicular and flushed to the plates).



The utility tool GeniEDXFImport can translate a DXF file with guiding geometry (poly-lines and poly-curves) to a neutral format (XML) that can be imported to GeniE. Observe that DXF poly-lines are converted to poly-curves or straight lines.

The picture to the right shows poly-curves imported into GeniE and a manual skinning of the hull between the selected guidelines has been performed.



The same may be achieved by transferring data from offset tables via the journaling system, i.e. offset tables are converted into guidelines subjected to skinning operations.

2.3 Load application strategies

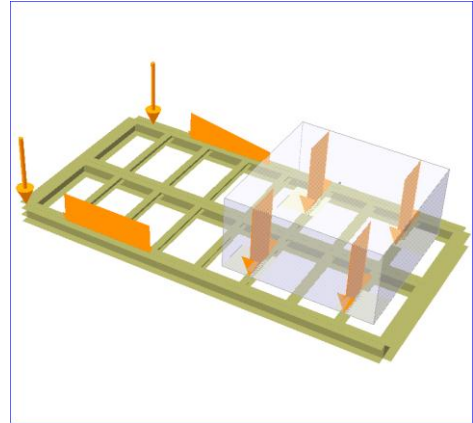
There are three main options for defining the loads. A commonality is that they are independent of the analysis model (or the mesh) you create since the “explicit” loads you apply will be converted to “applied” loads. In other words, you do not need to consider the mesh configuration when defining the loads.

- Manual load application – the loads you apply using typical point loads, line loads, pressure loads, equipment loads and acceleration loads (including gravity). The line loads and the pressure loads may be independent or associated with the structures – in the latter case the loads are moved when the structure is moved.
- Compartment loads – these are pressure loads that are generated from filling a compartment (full or partially) with liquid or solid content. The compartments are automatically generated provided the concept model has closed volumes.
- Rule based loads for bulk ships according to CSR (Common Structural Rules) – these are loads that are defined by Nauticus Hull in accordance with the CSR rules for bulk ships and automatically imported to the GeniE model.
- Creating loadcases for use by a panel model – these are load cases that needs to be defined for use in a hydrostatic or dynamic analyses. As such, they are not used in a design based load analysis from GeniE, but they are reference loadcases used by HydroD. Typically, the loadcases are used to describe the outer wet surface of a floating structure (by defining constant pressure acting on the outside) and the wet surfaces of the inside of compartments (by defining constant pressure acting on the inside). These attributes are automatically defined by GeniE.

Following are some examples given on each of these.

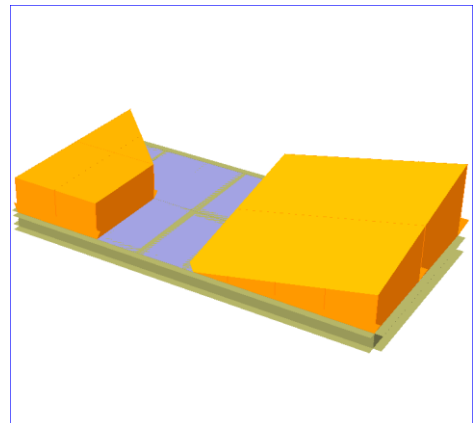
There are several ways of defining the “explicit” loads. The picture to the right shows that point loads may be applied at typical structural joints or at any position along a beam or stiffener. Similarly, line loads may be defined between structural connections or at any position on a beam.

Equipments may also be used to generate line loads where there is an intersection between a beam and the equipment footprints.

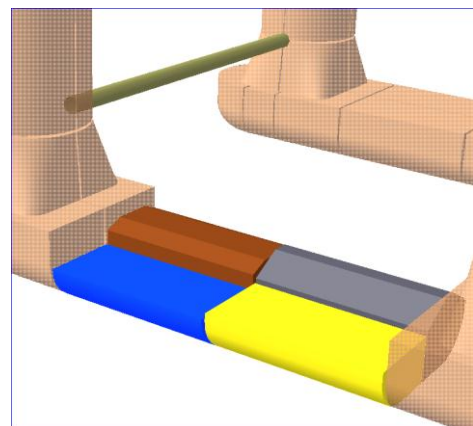


Pressure loads are applied to plate surfaces and they may be independent of the plates (or where they are typically split by stiffeners or plate seams). The example to the right shows that a pressure load can be applied anywhere on the plate (in this case the constant pressure load) or they may have a complex pressure definition (here shown as a 3-point varying pressure load).

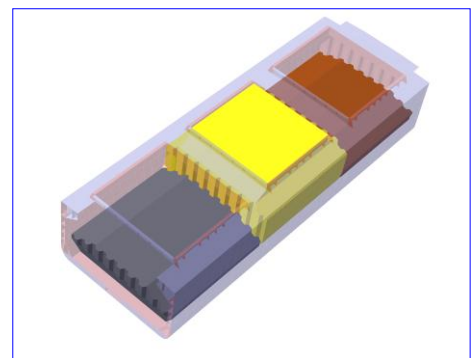
You may describe any line load or pressure load by defining your own load formula.



This example shows that four compartments have been filled with different content and filling heights. The contents used are water (blue), oil (yellow), bulk (brown) and ore (grey). As can be seen the top shape function of solid content can be modelled. For large compartments this can be of significant importance as the pressure is varying.



The picture to the right shows the mid-ship section of a bulk ship. The concept model has been defined to have bottom tanks, wing tanks and centre storage compartments. This information is automatically used by Nauticus Hull Rule Engine to compute the load cases according to the Common Structural Rules. The load case definitions as well as content and filling degree are seamlessly applied to the concept model to generate the loads being used in a design load based analysis approach.



2.4 Meshing strategies

When making analysis representations you need to create a finite element mesh. Depending on the type of analysis to be carried out you must decide a meshing strategy to ensure that the finite element model is fit for purpose. Typically, some of the strategies you should evaluate are:

- Mesh all structural parts in one go or specify a sequence on meshing the structural parts (mesh priorities)
- Mesh 2D structural parts first and keep the mesh for these when meshing the entire structure (a typical example of such is meshing web-frames first and lock the mesh for these)
- Make the mesh using different meshing algorithms. The options available are the *Sesam quad mesher* and an *advancing front mesher* (also known as a paver meshing algorithm). It is possible to mix the meshing techniques for the same concept model.
- Depending on the results you want to achieve you also need to decide on mesh density and type of finite elements to use. Normally, a finer mesh density gives more correct results and higher order elements (e.g. 8 node quad elements) are more accurate than first order elements (e.g. 4 node quad elements). However, the finer the model is the larger becomes the analysis to be carried out.
- Make the mesh using different mesh densities for various structural parts. It is not necessary to have the same fineness for the entire structure to achieve good results. Typically a joint needs fine mesh while it is sufficient to have a coarse mesh for a regular part of the structure – this will lead to satisfactory quality of the results while keeping the analysis model as small as possible. In these cases you can also decide the mesh growth rate.

As far as possible GeniE will always create a finite element mesh unless you have specified otherwise by setting thresholds for typically mesh corner angles or ratio aspects. The default values used by GeniE are mesh all in one go, use as large finite elements as possible and create first order elements.

The quality of the finite element results depends on the quality of the finite element mesh. There are basically two ways of controlling the mesh quality:

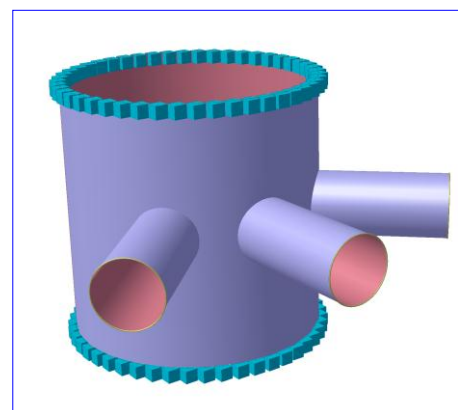
- By using the control mechanisms built into GeniE. In this case you specify the relevant check parameters if you want to over-ride the default settings
- By using your own experience or from calibration studies. Experienced engineers know how to set the mesh initially and also to adjust based on the results (typically by evaluating stress gradients after a finite element analysis). In cases where limited experience is available, calibration studies should be carried out to decide the initial mesh settings.

There are no default mesh settings that will guarantee the most correct or optimal analysis results. This is because the analysis results depend on both the structure configuration as well as the complexity of the loads. Therefore it is important that the user has sufficient control over the mesh generated and also means to adjust it so that the mesh is fit for purpose.

A small example of a tubular joint is used to exemplify how the mesh can be controlled and the significance of changing it.

The tubular joint has three incoming braces and the load case considered is a gravity load case.

Four different finite element models are analysed; the results are presented on the next page. Notice, this example is meant as an example on how important the mesh settings can be for the analysis results. For another load case, the same mesh settings may give other relative differences for the analysis results.



Mesh settings:

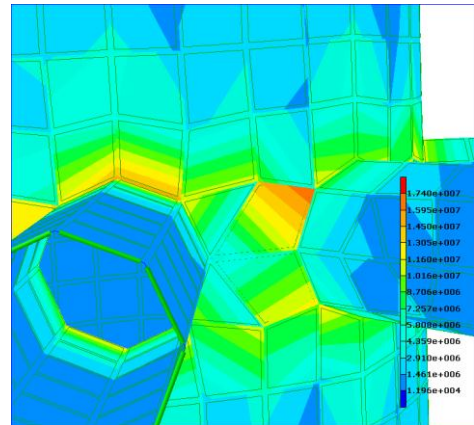
- First order finite elements, advancing front quad mesher, 1 m characteristic mesh size

Analysis characteristics:

- 2900 DOF, 2 sec elapsed CPU on reference laptop computer

Analysis results:

- Maximum Vonmises stress $1.740\text{E}07$ Pa.
- Stress gradients to high to use results in critical region
- For a global analysis perhaps adequate to use



Mesh settings:

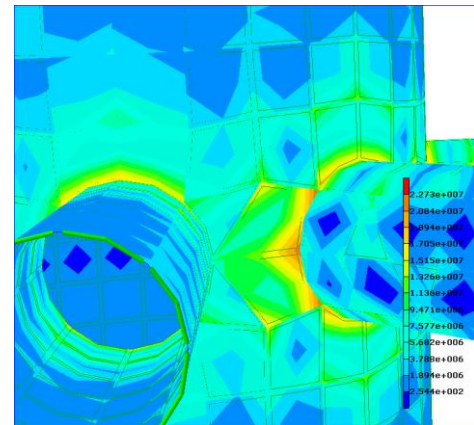
- Second order finite elements, advancing front quad mesher, 1 m characteristic mesh size

Analysis characteristics:

- 8500 DOF, 3 sec elapsed CPU on reference laptop computer

Analysis results:

- Maximum Vonmises stress $2.273\text{E}07$ Pa.
- Stress gradients to high to use results in critical region
- For a global analysis perhaps adequate to use



Mesh settings:

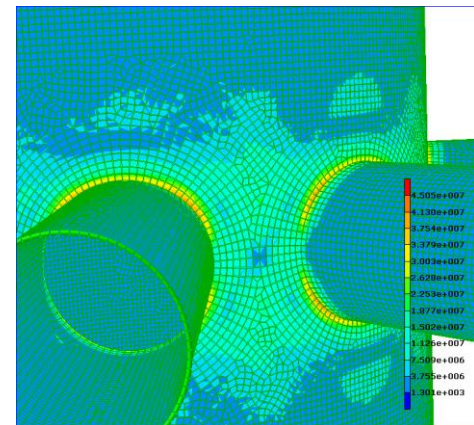
- First order finite elements, advancing front quad mesher, 0.1 m characteristic mesh size

Analysis characteristics:

- 252.000 DOF, 187 sec elapsed CPU on reference laptop computer

Analysis results:

- Maximum Vonmises stress $4.505\text{E}07$ Pa
- Stress gradients look reasonable
- Notice that fatigue analysis requires mesh size typically same as the plate thickness



Mesh settings:

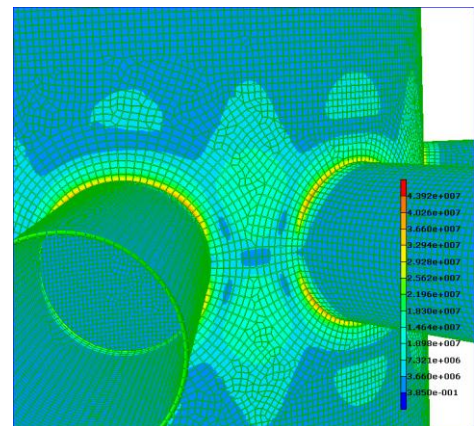
- Second order finite elements, advancing front quad mesher, 0.1 m characteristic mesh size

Analysis characteristics:

- 755.000 DOF, 1752 sec elapsed CPU on reference laptop computer

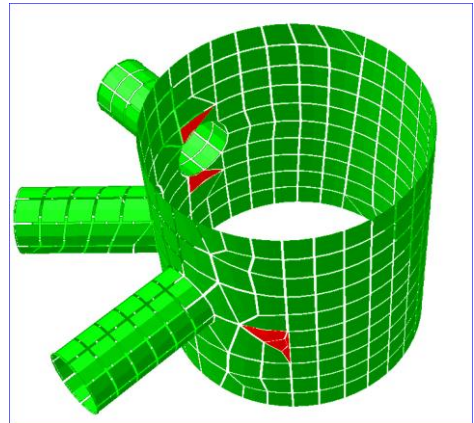
Analysis results:

- Maximum Vonmises stress $4.392\text{E}07$ Pa
- Stress gradients look even more reasonable
- Notice that fatigue analysis requires mesh size typically same as the plate thickness



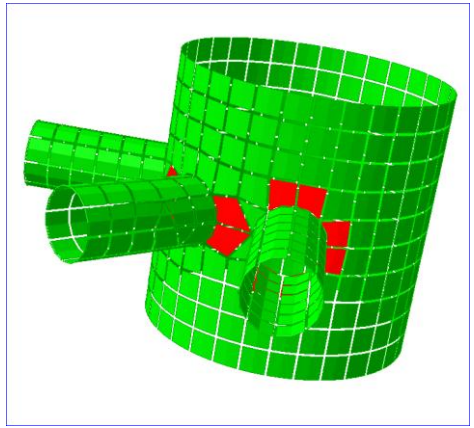
For a structure like this it is also important which meshing algorithm is being used. The *Sesam quad mesher* in GeniE is intended for relatively regular structures, typically a ship hull, a TLP, a spar or semi-submersible. In many cases the Sesam quad mesher will give a satisfactory mesh for complex structures with additional mesh control.

In the example to the right the Sesam quad mesher has been used without any mesh control. The highlighted areas indicate where mesh corner angles are above 150 degrees and the relative Jacobi factor is larger than 4.0.



For structures like this the mesh quality normally becomes much better when using the *advancing front quad mesher*. Without any additional mesh controls, the mesh is significantly improved. There are no mesh corner angles above 115 degrees in this case, and there are no finite elements having a relative Jacobi factor larger than 1.7.

The finite elements having a factor above 1.5 are shown to the right – in other words a significant improvement just by changing the mesh algorithm.



3. PLATE & SHELL STRUCTURES

The following Chapters describe how to do the various tasks to model and analyse a plate/shell structure using a design load based analysis approach. This means that the loads are assumed to be applied and computed in GeniE as opposite to a direct analysis approach where some of the governing loads are defined by the hydrodynamic module HydroD.

The content of this Section have been organized so that they follow a typical working procedure

- Set up the design premise by defining the section profiles, materials and plate thicknesses
- Create guiding geometry for use when making plates and stiffeners. Notice that importing data from other sources to generate guiding geometry is dealt with at the end of Chapter 3. The Section describing guiding geometry also documents operations common for modelling of other objects, typically graphic handling
- Make plate/shell structure including stiffeners by referencing guiding geometry. It is also possible to re-use data from other sources to directly create the structure.
- Apply the loads to the model – these may be manually defined or computed from compartment filling.
- Define the boundary conditions
- Make and control the finite element mesh
- Run analysis and look at results.

3.1 The design premise

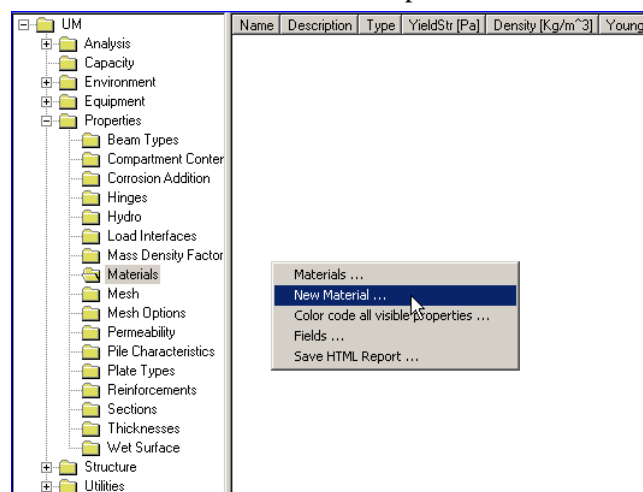
Prior to modelling you may define which plate thicknesses, section profiles and material types you want to use. Observe that these properties can be defined and modified at any time and applied to the model when needed. If you modify a property the changes automatically apply to those structural concepts the properties are assigned to. In other words, you can change properties assigned to a concept at any time.

You can define your own property values, you can use property values part of GeniE libraries or you can create your own libraries. These methods are described in the following Sections.

3.1.1 Material properties

Materials can be defined from the command *Edit/Properties/Materials* or from **RMB** in the browser area (see picture below). There are two types of materials that can be defined: Linear Isotropic Material and Isotropic Shear Material.

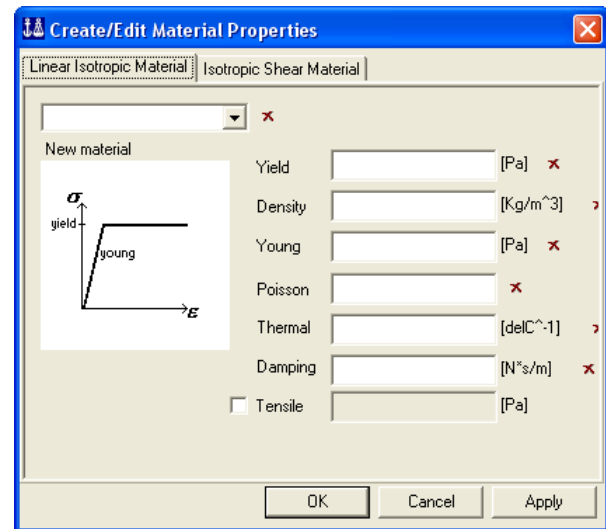
Right click on folder “Materials” or in right browser pane to access the menu for defining materials.



The linear isotropic material is the most commonly used material property for use in linear structural analysis. Notice that you need to fill in all values as there are no default values.

The yield value is used when code checking beams.

In case you want to create a model in GeniE for later use in Usfos, you can define the tensile values as well. For linear structural analysis this value is not used.

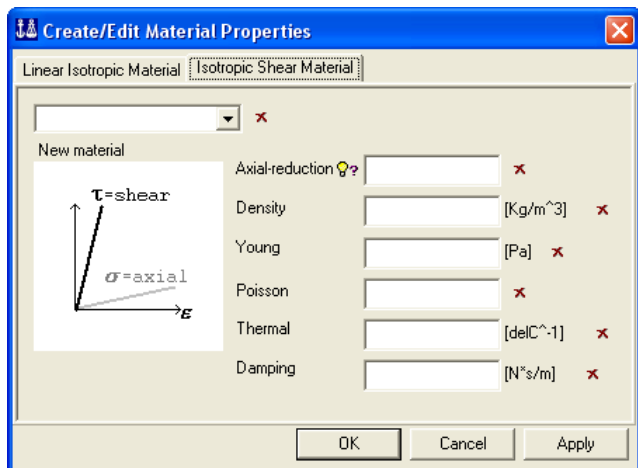


The isotropic shear material is used when you want to differentiate between the axial and shear part of the material. You can reduce the axial components by defining an “Axial-reduction” factor.

Axial Reduction

- Axial reduction (e.g. 100 means 1/100 axial stiffness)

Typically, this material type is used on a plate part of a wall where the vertical axial forces are supposed to be carried by the columns.



You can also import a material library from c:\Program Files\DNVS\GeniE\Libraries\Material_library.xml. The path name assumes you have installed the program using the default installation set-up. The library is imported by using the command **File/Import/XML Concept Model**.

The library contains 71 linear isotropic materials; these are documented in Appendix B.

UM	Name	Description
Analysis	ASTM_A285_Grade_C	Material, lin. isotropic, E=2.1
Capacity	ASTM_A36	Material, lin. isotropic, E=2.1
Environment	ASTM_A131_Grade_A	Material, lin. isotropic, E=2.1
Equipment	ASTM_A131_Grade_B_D	Material, lin. isotropic, E=2.1
Properties	ASTM_A516_Grade_65	Material, lin. isotropic, E=2.1
Beam Types	ASTM_A573_Grade_65	Material, lin. isotropic, E=2.1
Compartment Center	ASTM_A709_Grade_36T2	Material, lin. isotropic, E=2.1
Corrosion Addition	ASTM_A131_Grade_CS_E	Material, lin. isotropic, E=2.1
Hinges	ASTM_A572_Grade_42	Material, lin. isotropic, E=2.1
Hydro	ASTM_A572_Grade_50	Material, lin. isotropic, E=2.1
Load Interfaces	API_Spec_2MT2_Class_C	Material, lin. isotropic, E=2.1
Mass Density Factor	API_Spec_2W_Grade_42	Material, lin. isotropic, E=2.1
Materials	ASTM_A992	Material, lin. isotropic, E=2.1
Mesh	API_Spec_2MT1	Material, lin. isotropic, E=2.1
Mesh Options	API_Spec_2MT2_Class_B	Material, lin. isotropic, E=2.1
Permeability	ASTM_A709_Grade_50T2_50T3	Material, lin. isotropic, E=2.1
Plate Characteristics	ASTM_A131_Grade_AH32	Material, lin. isotropic, E=2.1
Plate Types	ASTM_A131_Grade_AH36	Material, lin. isotropic, E=2.1
Reinforcements	API_Spec_2H_Grade_42	Material, lin. isotropic, E=2.1
Sections	API_Spec_2H_Grade_50	Material, lin. isotropic, E=2.1
Thicknesses	API_Spec_2W_Grade_50T	Material, lin. isotropic, E=2.1
Wet Surface	API_Spec_2Y_Grade_42	Material, lin. isotropic, E=2.1
Structure	S_460_N_NL	Material, lin. isotropic, E=2.1
Utilities	API_Spec_2W_Grade_50T	Material, lin. isotropic, E=2.1

3.1.2 Section properties

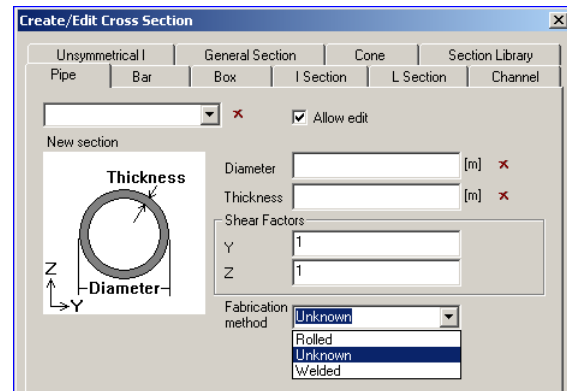
Section properties can be defined from the command *Edit/Properties/Section* or from **RMB** in the browser area.

There are nine types of sections (or profiles) that can be defined – Pipe, bar, Box, I-section, L-section, Channel, Unsymmetrical I, General Section and Cone.

The Cone property is used in connection with conical transitions during segmented beam modelling.

The shear factors will reduce the shear contribution. The default value is full contribution from shear.

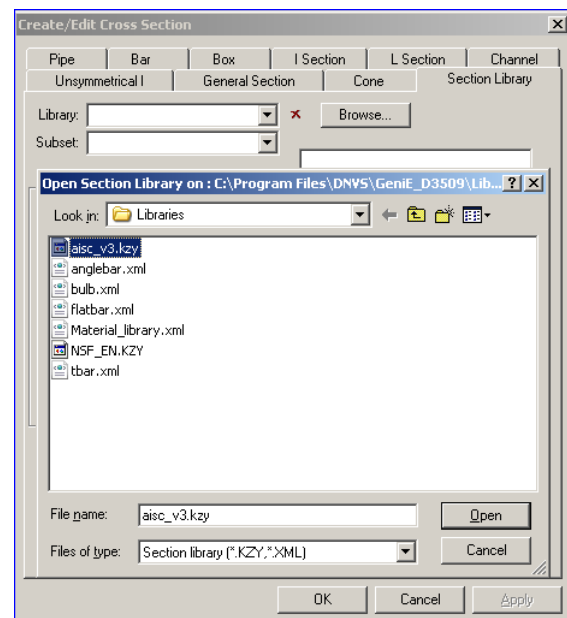
The fabrication method has impact on the beam utilisation factors as these attributes are used during code checking. The default value is “Unknown” fabrication method.



You can also import section properties from a section library. Click on the tab “Section Library” and “Browse” to see which libraries are available. There are close to 6000 standard profiles to select from the

- AISC library (SI and US)
- NSF (Eurocode) library
- An anglebar library
- A bulb library
- A flatbar library
- A tbar library

You may create your own section libraries and get access to these from the “Section Library” tab by using the same approach as explained in previous Chapter.



The theory used when computing the derived properties is listed on GeniE’s help page under “Reference documents”.

3.1.3 Thickness properties

Thickness properties can be defined from the command *Edit/Properties/Thickness* or from **RMB** in the browser area.

There are no libraries (or pre-defined) thicknesses in GeniE, but it is easy to create your own library set by using the XML file approach as explained above or by defining and re-using a journal file (typically a “My_thickness.js” file).



3.2 Create guiding geometry

Guiding geometry is used as reference when defining structural concepts or boundary conditions. The guiding geometry may be a point, a straight line, a curved line or a so-called guiding plane.

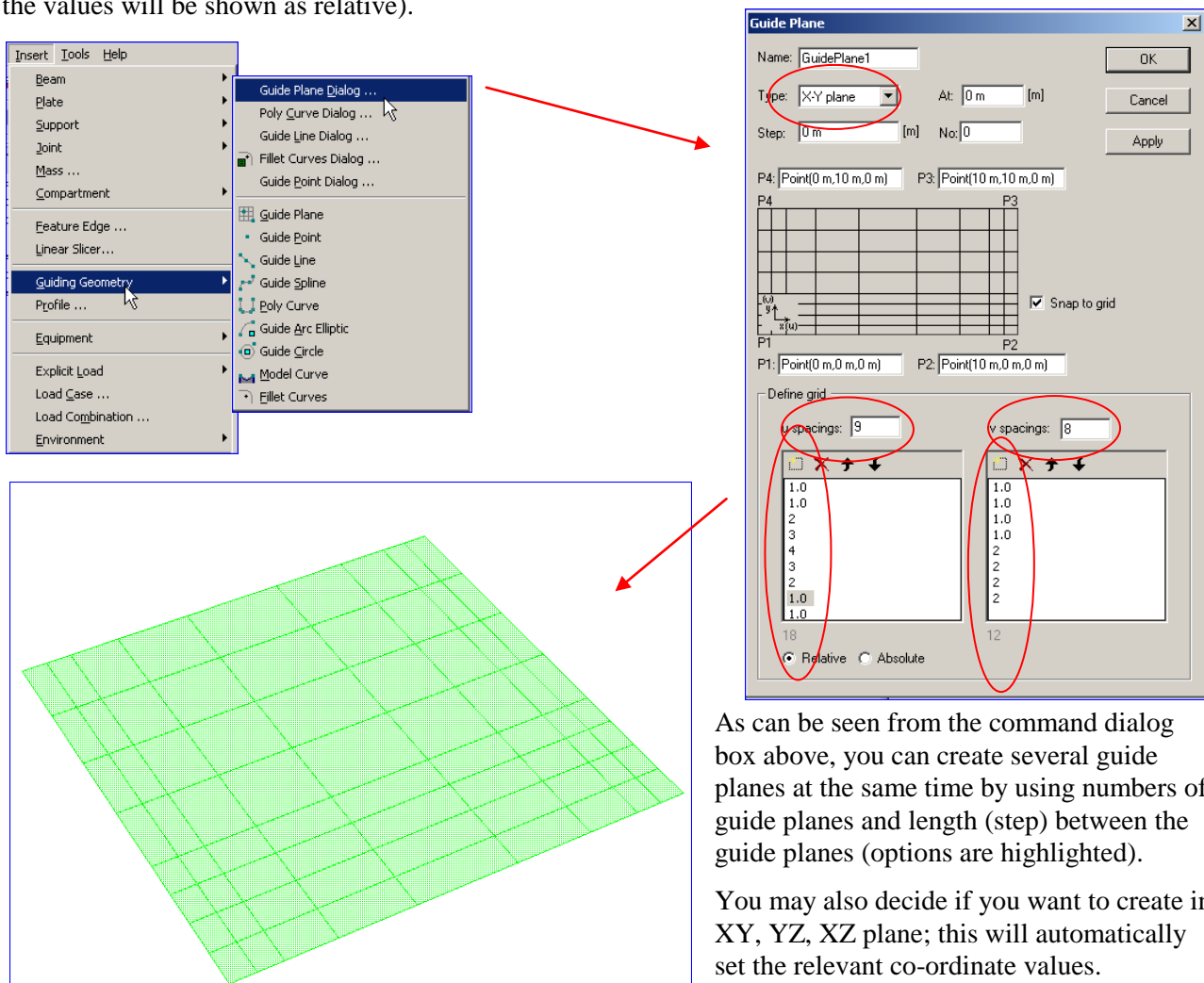
You can create guiding geometry by defining it manually, from the topology of an existing concept model (e.g. along the edge of a plate) or from importing data from external sources (typically a CAD model or an offset table). These methods are described in the following; please notice that importing data is documented later in this Section.

For all methods apply that when guiding lines intersect they will automatically create a snap point for reference.

3.2.1 Guide plane

A guide plane may be defined from the *Insert/Guiding Geometry/Guide Plane Dialog*. A guide plane is often the start of modelling tasks, since it is used to create other guiding geometry, or form the basis for structural parts. Typically a guide line or a beam can be defined by referring to two positions on the guide plane. Similarly a plate may be defined by referring to four positions.

In the example below a guide plane has been defined in the XY-plane with 9 and 8 spacings in the u and v directions. The length of each spacing is in this case defined using a relative value. You can also use absolute values for the spacing (notice that if you later select the guide plane, **RMB** and select *Properties*, the values will be shown as relative).

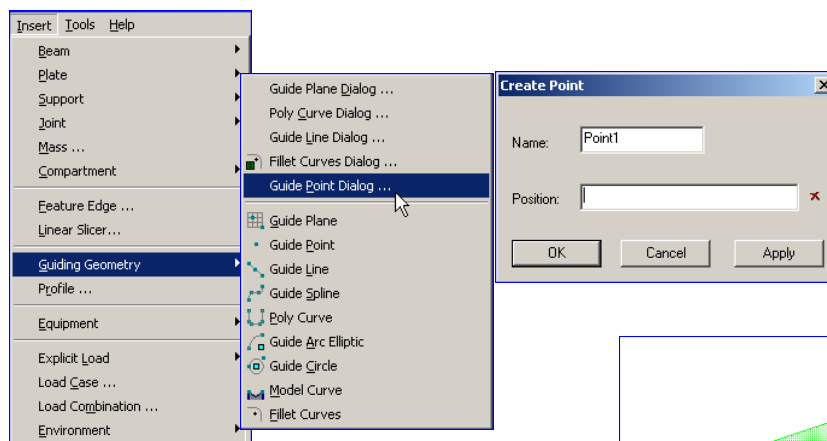


If you have existing guiding geometry (like guiding points) you can create a guide plane by clicking on 4 snap points. You do this from the **Insert/Guiding Geometry/Guide Plan**. In this case a guide plane is defined between the 4 reference points with 4 equal spacings in u and v direction. You may modify number of spacings and their relative lengths by selecting the guide plane, **RMB** and *Properties*. In this case the automatic naming schema is used to define the names of each guide plane you create (the default name is GuidePlane*).

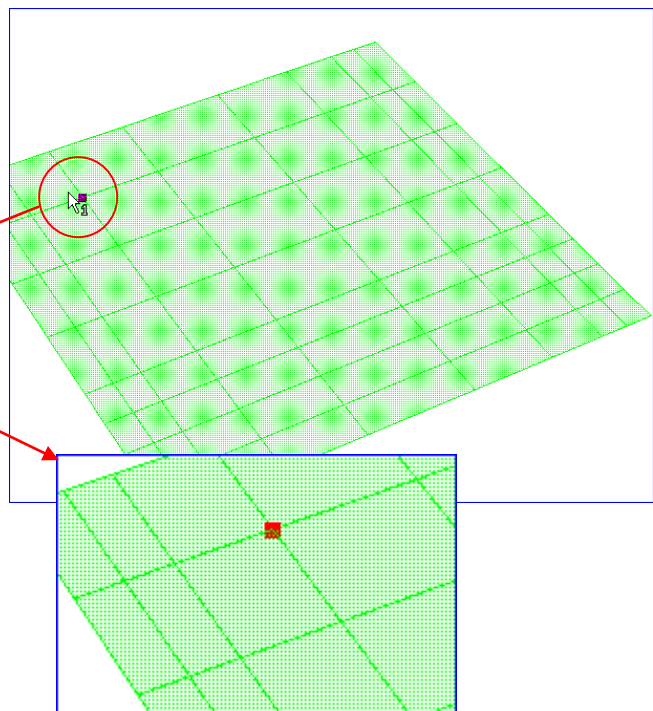
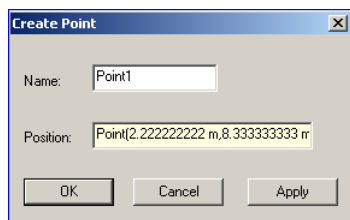


3.2.2 Guide point

Below is shown how to define a guide point – it may be defined from the **Insert/Guiding Geometry/Guide Point Dialog**.



You may now specify a co-ordinate value or move the mouse to the graphical window. By clicking a reference point the co-ordinate values are automatically found from the reference point.

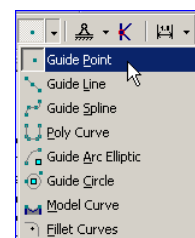


The new guiding point is shown to the right.

You may also define a guide point by clicking on 1 snap point. You do this from the **Insert/Guiding Geometry/Guide Point** (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each guide point you create (the default name is Point*).

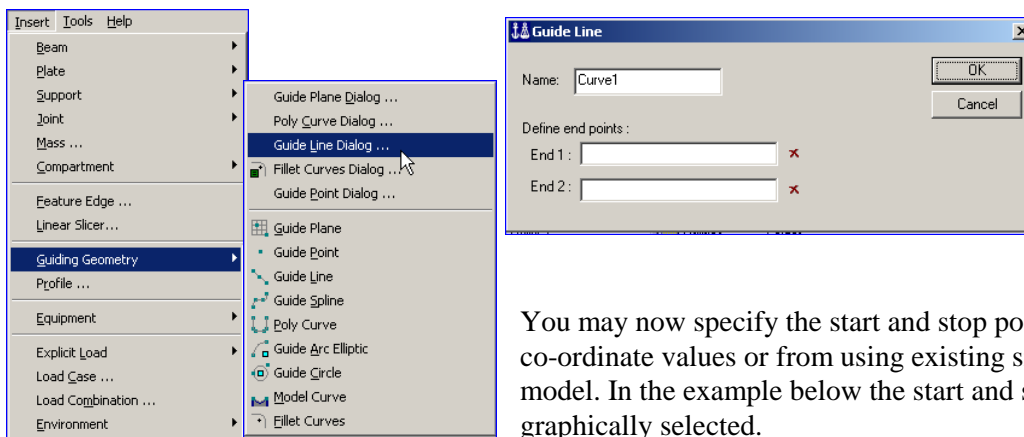
Typically, if you repeat this process 3 times (with different co-ordinate values) you will have 4 guide points to draw a poly-curve or guide spline between.

The guide point may be modified from selecting the line, **RMB** and *Properties*.

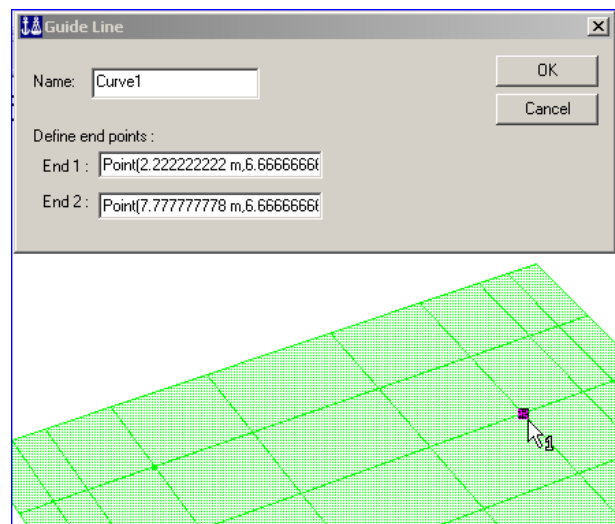
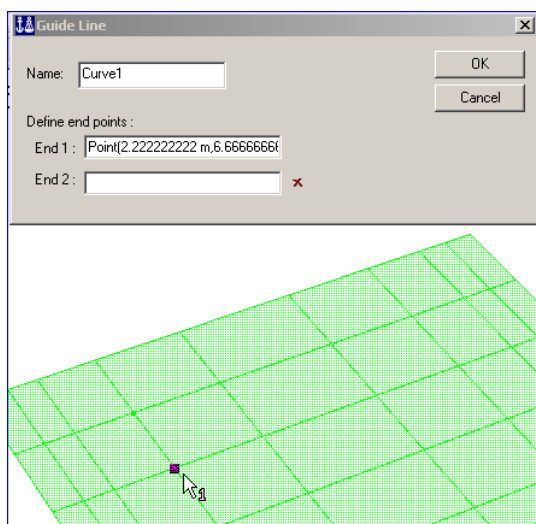


3.2.3 Guide line

A guide line may be defined from the *Insert/Guiding Geometry/Guide Line Dialog*.



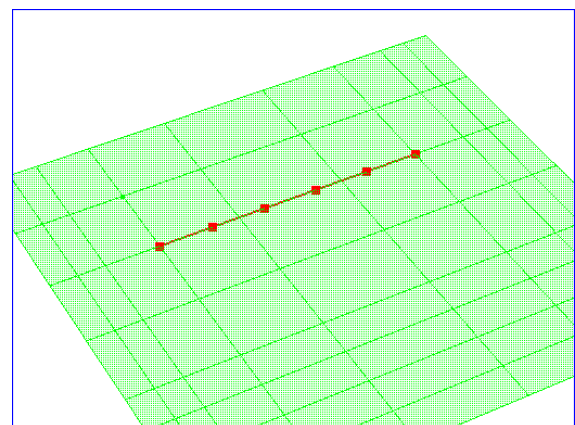
You may now specify the start and stop positions by defining the co-ordinate values or from using existing snap points in your model. In the example below the start and stop positions are graphically selected.



A straight guide line is now created as shown to the right. Per default the guide line has 4 internal snap points; these may be modified in the command window (or by use of the scripting language) using the following command:

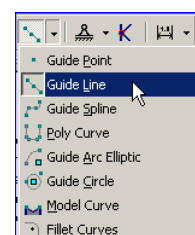
```
Curve1.spacings(Array(1,1,2,3,4,4,3,2,1,1));
```

This will modify the guide line Curve1 to have 10 segments (11 snap points) with relative lengths as shown in the command above.



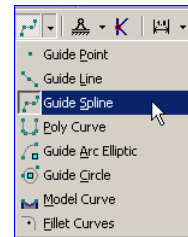
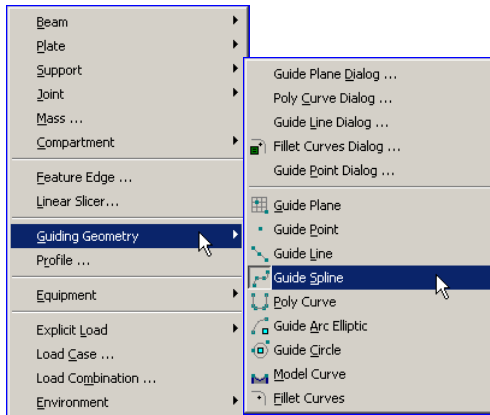
You may also define a guide line by clicking on 2 snap points. You do this from the *Insert/Guiding Geometry/Guide Line* (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each guide line you create (the default name is Curve*).

The guide line may be modified from selecting the line, **RMB** and *Edit GuideLine*.



3.2.4 Guide Spline

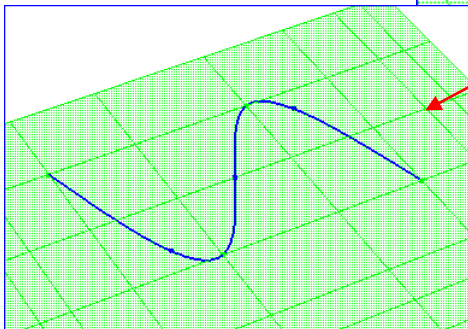
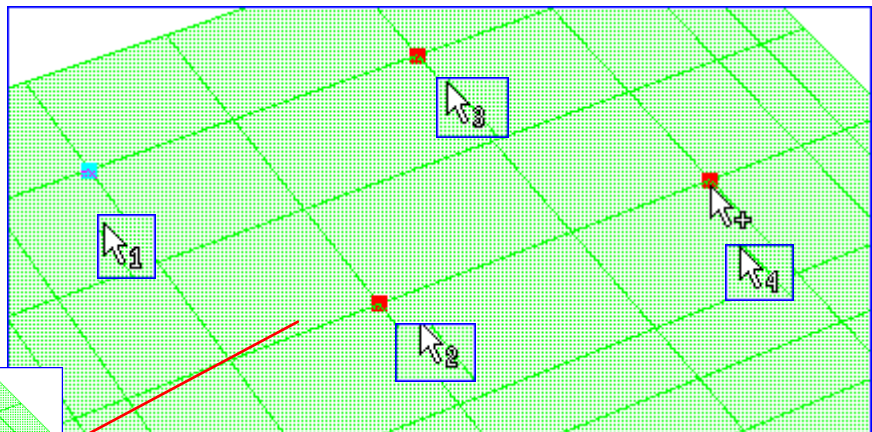
Guide splines are inserted by referencing existing snap points. The command is activated from **Insert/Guiding Geometry/Guide Spline** (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each guide spline you create (the default name is Curve*).



The guide spline is now computed based on number of positions you add. There may be several positions and you stop the input sequence by a double click

In the example below, the guide spline is defined from 4 snap points (i.e. click four times and a double click on the last position).

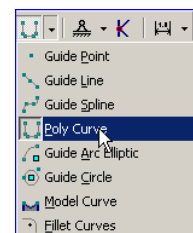
When you want to use more than 4 points the mouse indicator will look like:



3.2.5 Poly-curve

Guide splines are inserted by referencing existing snap points. The command is activated from **Insert/Guiding Geometry/Poly Curve** (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each poly curve you create (the default name is Curve*).

A poly curve is a curved built up of straight segments and spline segments. It is characterised with a constant tangential along its line.

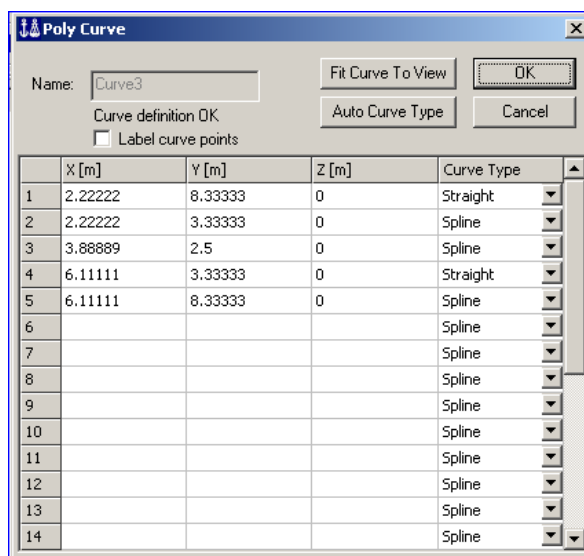
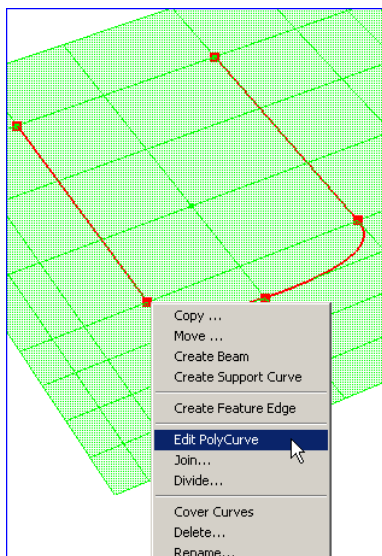
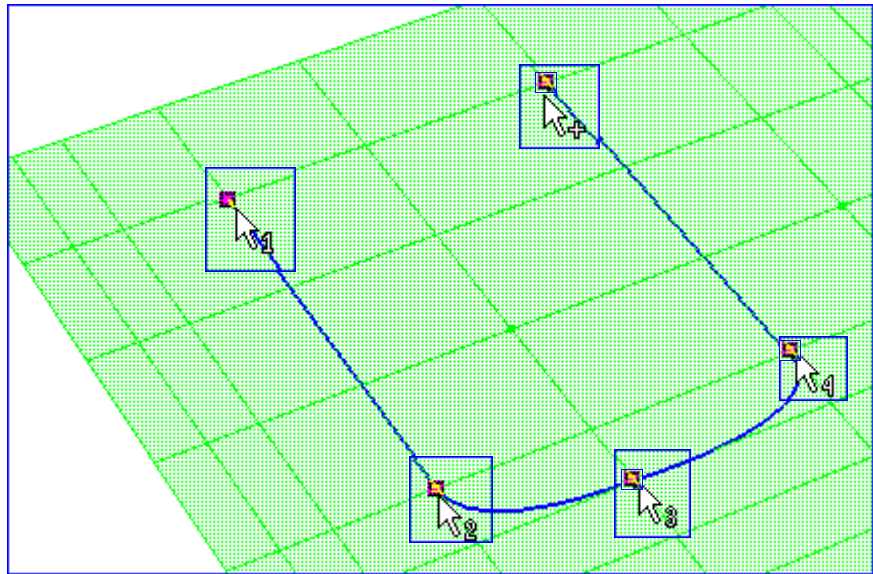


You may use multiple points when defining the curve and the curvature is controlled by the number and position of the points along the line.

The example to the right shows a poly curve generated by 5 points (double click on the last point to stop the curve).

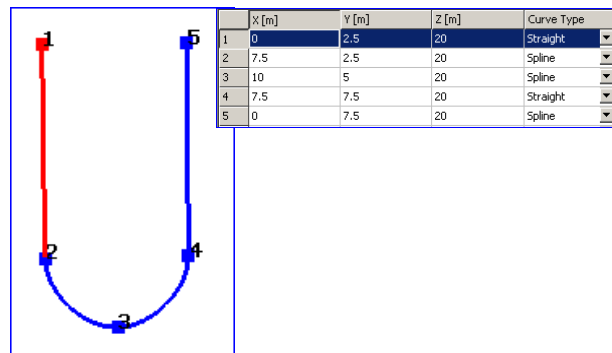
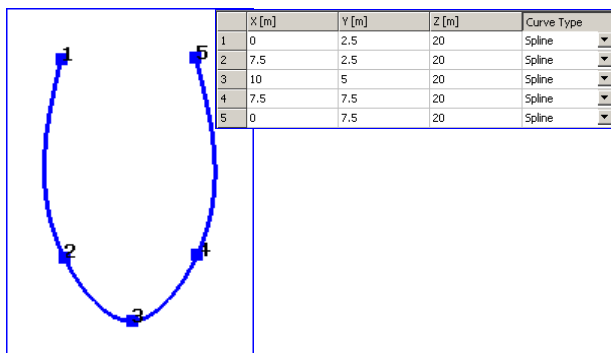
You may edit the curve by changing the co-ordinates or the curve type.

The curve may be edited by selecting the curve, **RMB** and *Edit Polycurve*.



The “Label curve points” adds labels to each of the points along the poly curve.

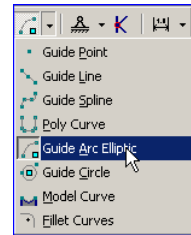
The “Auto Curve Type” will seek to make a poly curve with as many straight curve segments as possible. Typically a curve with spline segments only becomes a curve with 2 straight and 2 spline segments



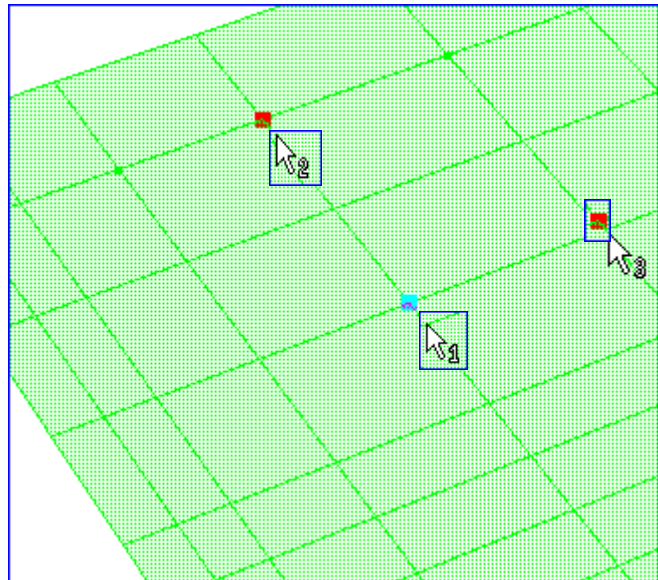
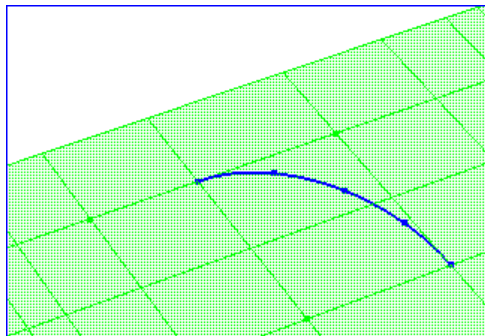
3.2.6 Guide arc elliptic

A guide arc elliptic is inserted by referencing existing snap points. The command is activated from **Insert/Guiding Geometry/Guide Arc Elliptic** (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each elliptic arc you create (the default name is Curve*).

A guide arc elliptic is a curved built up by referring to 1) the origin, 2) the start and 3) stop position of an arc. A 1/4 circle may be generated when the start and stop positions have the same radius and are perpendicular to the centre of the circle.

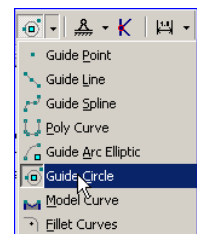


The modelling sequence to the rights will give an arc as shown below.



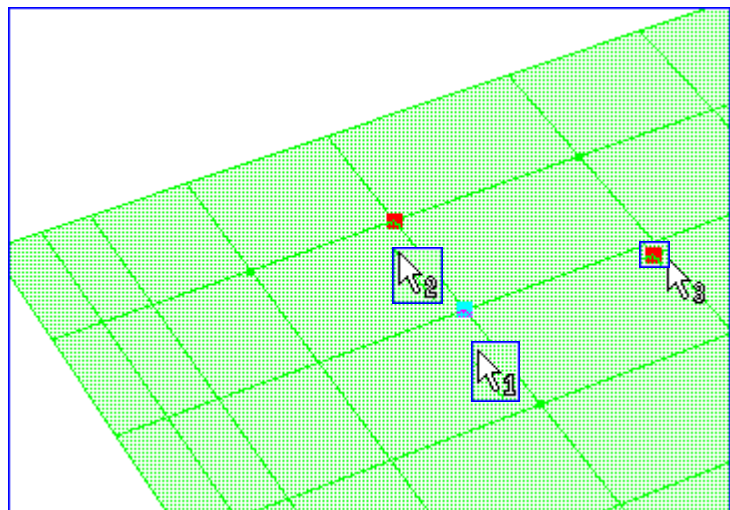
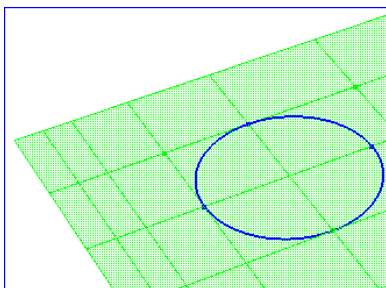
3.2.7 Guide circle

A guide circle is inserted by referencing existing snap points. The command is activated from **Insert/Guiding Geometry/Guide Circle** (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each guide circle you create (the default name is Curve*).



A guide arc elliptic is a curved built up by referring to 1) the origin, 2) the radius and 3) a point to define the circle plane.

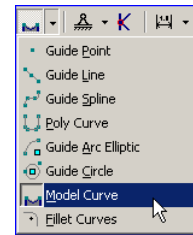
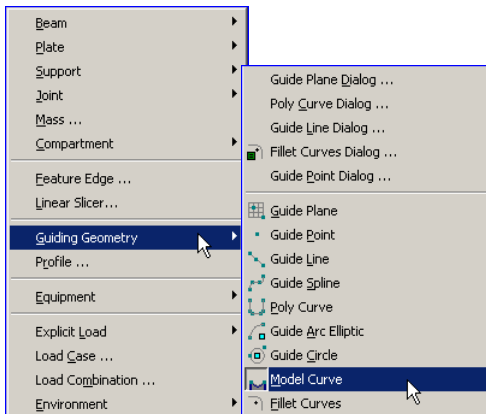
Since points 1, 2 and 3 form a horizontal plane, the circle will be horizontal with the origin in point 1 and a radius equal to the distance between point 1 and 2.



3.2.8 Model curve

Model curves may be inserted between two points or along a topology line. The difference between a model curve and a straight guiding line is that the model curve will follow the surface curvature.

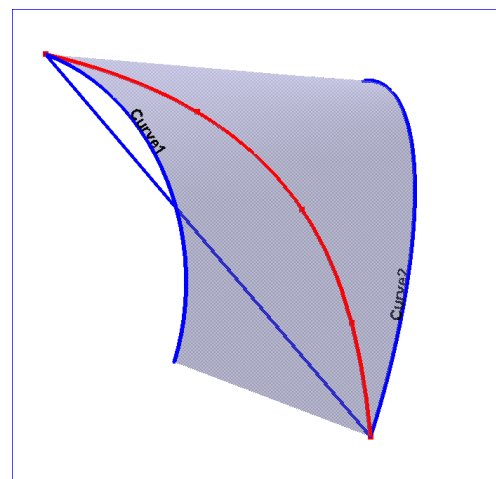
The command is activated from **Insert/Guiding Geometry/Model Curve** (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each model curve you create (the default name is Curve*).



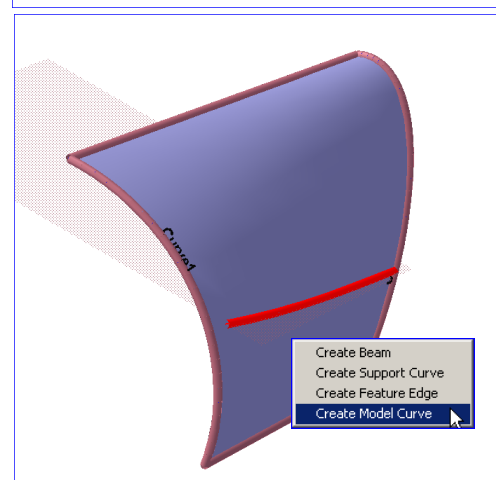
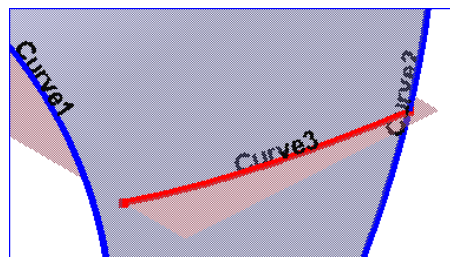
The model curve can be inserted by referencing two existing snap points (a point or vertices along a guide line or curve) or by assigning the model curve to a topology line. Both options are shown below.

In the example to the right, the model curve (highlighted in red) has been inserted between two vertices on curves (Curve1 and Curve2) used to define a curved surface (surface definitions are documented later in this user manual). In this case vertices at curve ends were selected. A guide line has also been inserted between the same vertices, and as can be seen the model curve follows the surface curvature, while the guide line is straight between the vertices.

The model curve can be used to e.g. make new plates, split the plate or to add a beam, boundary conditions and mesh control.



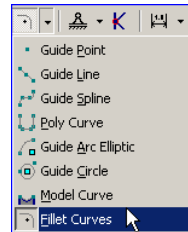
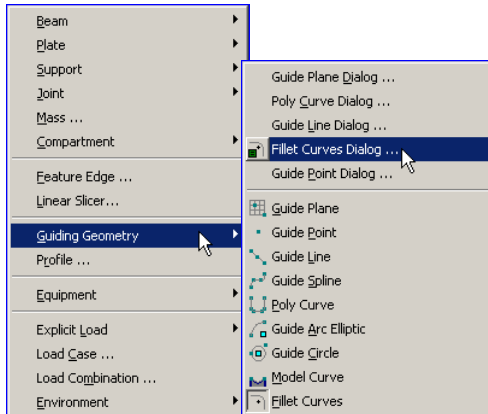
When assigning a model curve to a topology line it is necessary to select the topology line. This can be done by double-clicking a plate. The picture to the right shows the topology lines (connectivity lines between objects) for the curved surface intersected by a horizontal plate. Simply right click the topology line to access the command to insert a model curve (in this case Curve3). Remember to double-click the plate to get back to normal modelling view.



3.2.9 Fillet curve

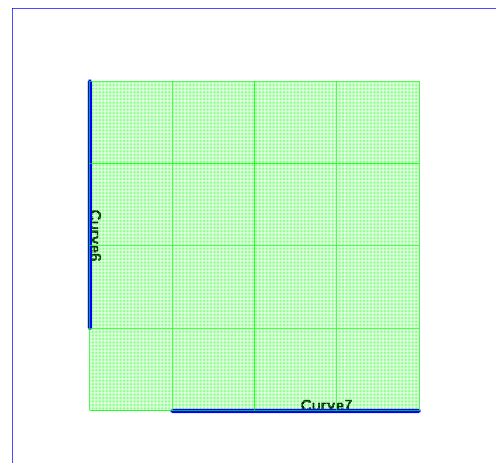
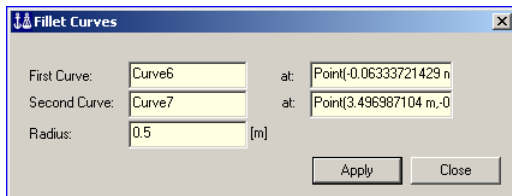
Fillet curves are used to define the curvature between two straight lines. Fillet curves are typically used when defining cut-outs in surfaces like for example man-holes or other openings. When fillet curves are used, the two straight lines in question and the fillet curves are automatically joined to a composite curve (see also section on joining curves).

Fillet curves are inserted by referencing guiding lines. The command is activated from **Insert/Guiding Geometry/Fillet Curves Dialog** (or from the tool button as shown to the right). The automatic naming schema is used to define the names of each composite curve you create (the default name is Curve*).

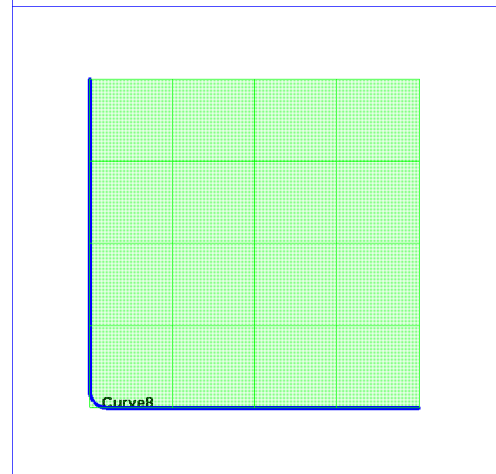
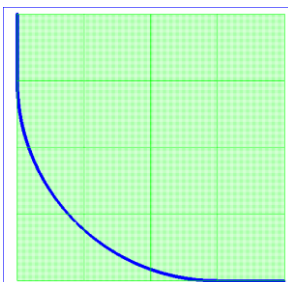


The composite curve is computed based on the orientation of the two straight lines and the radius to be used. If you use the option **Fillet Curves** the minimum possible radius without trimming any lines will be used as default. In the **Fillet Curves Dialog** you also have the option of specifying the radius.

In the example to the right there are two curves perpendicular to each other. When using the option Fillet Curves Dialog, referencing to the lines (either by giving the name or by finding the name from the graphical window) and specifying the radius the composite curve is generated. In this case Curve6 and Curve7 are used with a radius of 0.5m and the composite curve Curve8 is generated. As can be seen the straight lines have been extended to ensure continuation between straight and curved lines.

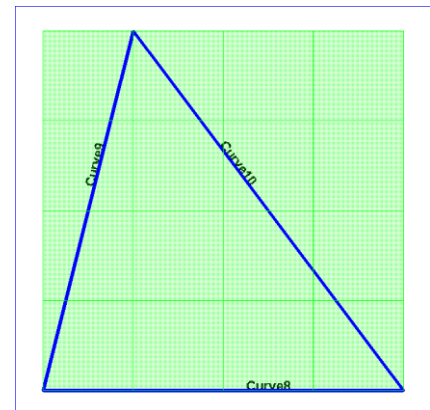
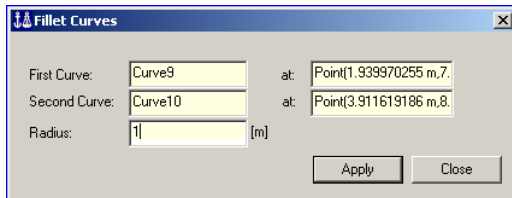


When using a value for radius larger than the minimum to fit the lines and the curve, the straight lines will be trimmed so that there is a continuation between the straight and curved parts. This is shown in the example below where radius is set to 7.5m

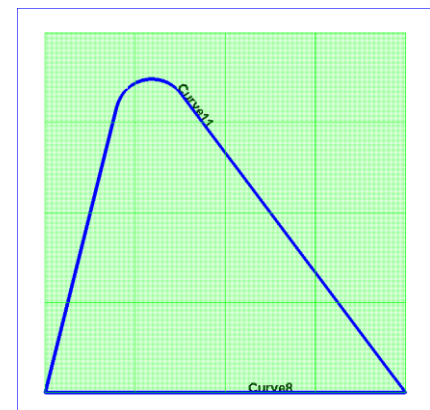
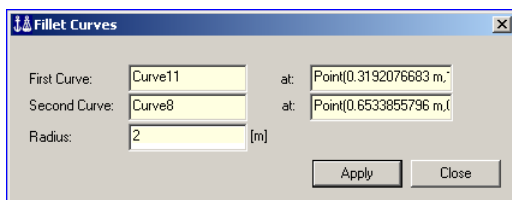


To make the lines shown to the right with rounded parts, it is necessary to do the fillet curve operation three times. You can start at any position to complete the operation.

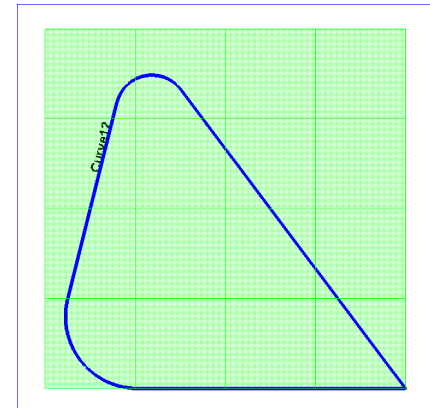
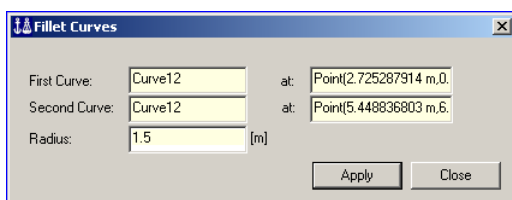
In this case Curve9 and Curve10 are made composite curve using a fillet radius of 1m; Curve11 is generated.



The next operation is now to select Curve11 (and clicking on it on the left most side) and Curve8. In this case a fillet radius 2m is used and Curve12 is generated.

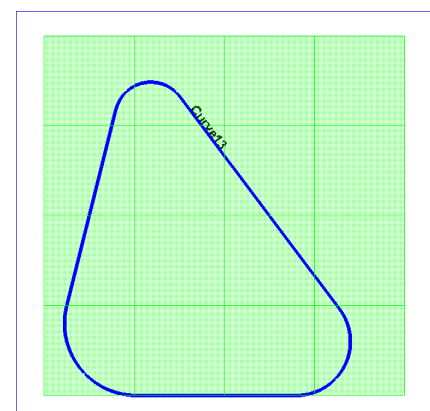
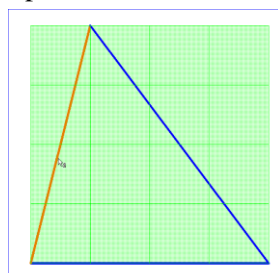


The final task is now to add a fillet radius to the triangular part. Select Curve12 somewhere at the left bottom part and then Curve12 at the upper right part. A fillet radius 1.5m gives the final composite curve Curve13.




This technique can be used to define the profile of holes and other cut-outs.

The same can be achieved by using the option **Fillet Curves**. In this case you type in the radius when you are in graphics mode before you hit the Enter button (or carriage return). The lines are highlighted in orange when selecting them. In the example below the first line has been selected.

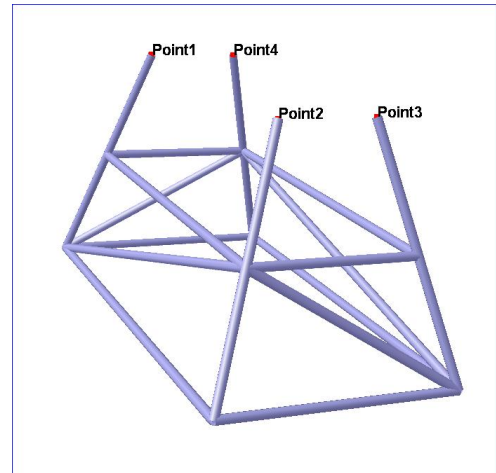
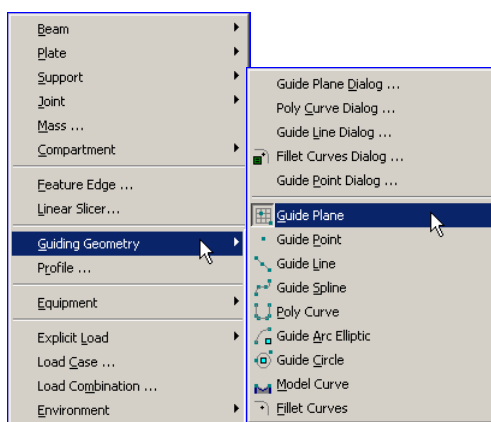


3.2.10 Model guiding geometry using existing snap features

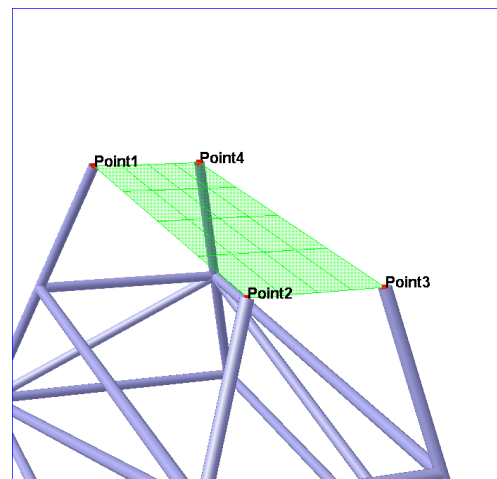
Common for all modelling in GeniE is that you can build on existing snap points like for example a beam end, a plate corner or vertices along a guiding line (see Section in the following for a definition of vertices). Guiding geometry can also be inserted using such techniques. An example of this can be when inserting a guide plane between four existing snap points.

In case you want to insert a guide plan between the points as shown to the right, a quick way is to insert the guide plane from the tool button 

Alternatively you can do the same from



By clicking the four points sequentially (Point1 -> Point4) a guide plane is defined as shown to the right. For a guide plane, the numbers of spacings in u and v directions are automatically set to 4 equal spacings. You may modify this by selecting the guide plane, **RMB** and choose *Properties* (see Sections in the following to modify).

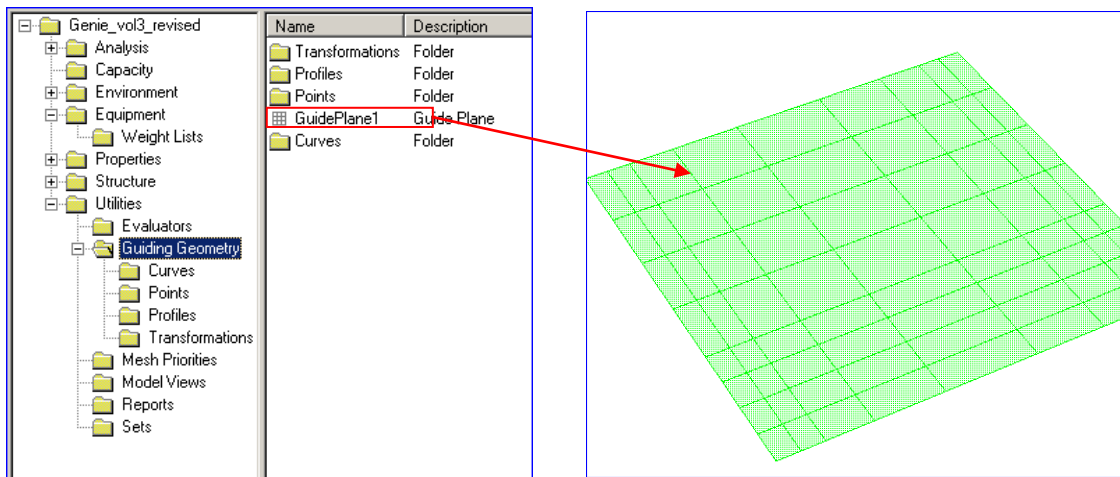


3.2.11 Find, select and display guiding geometry

There are two ways of finding and selecting guiding geometries (or other objects like beams, plates, equipments, boundary conditions and so on).

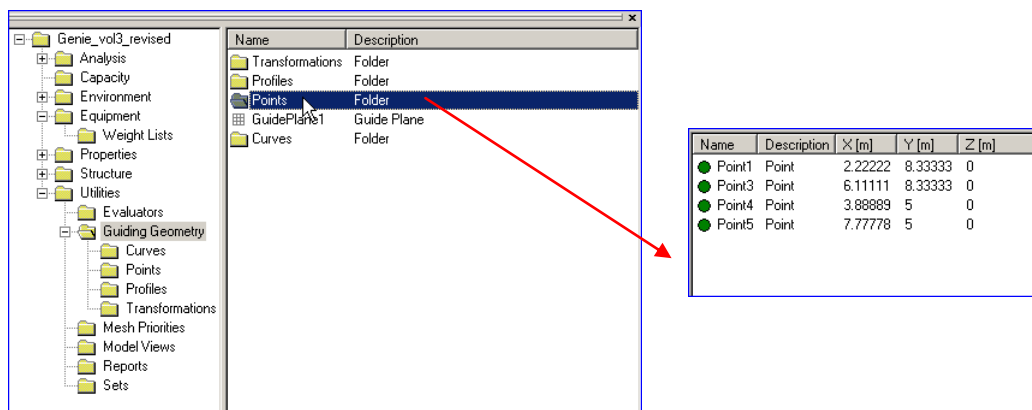
When selecting from the graphic window you click on the object you want to select. It is now highlighted and if you have the right browser pane open, the same object is highlighted in the browser.

It is also possible to select from the graphics. Guiding geometry is found under the browser tab *Utilities* and then *Guiding Geometry*. You can now find all of your guiding geometry whether it is a guide plane, a guide point, a guide curve and so on. If you click on the GuidePlane1 as shown below, the guide plane in the graphic window will be highlighted.



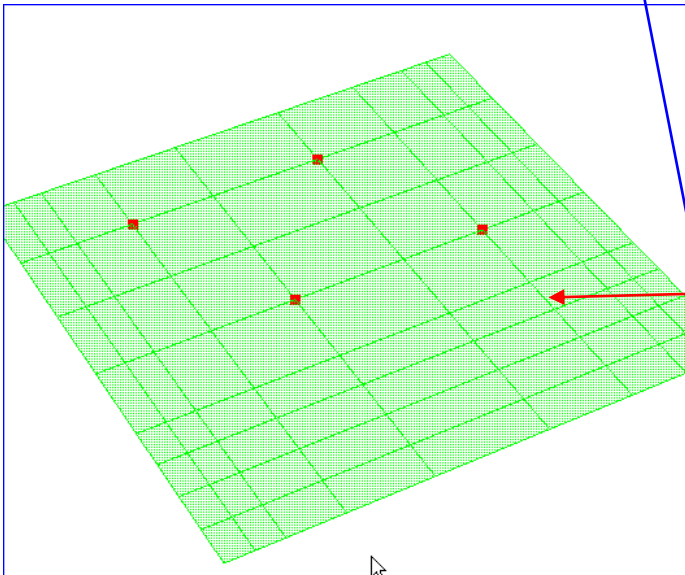
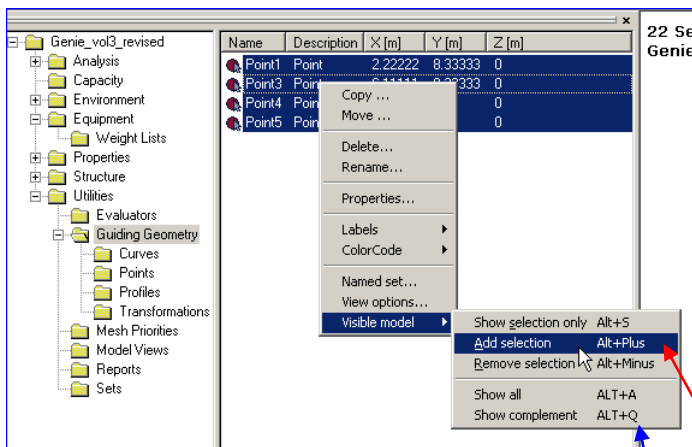
When doing this in GeniE, the actual browser name and object are highlighted (not shown here because of visibility).

Observe that it is also possible to drill down the browser also from the right pane browser window. If you e.g. double click the browser tab *Points* below you will open the content.

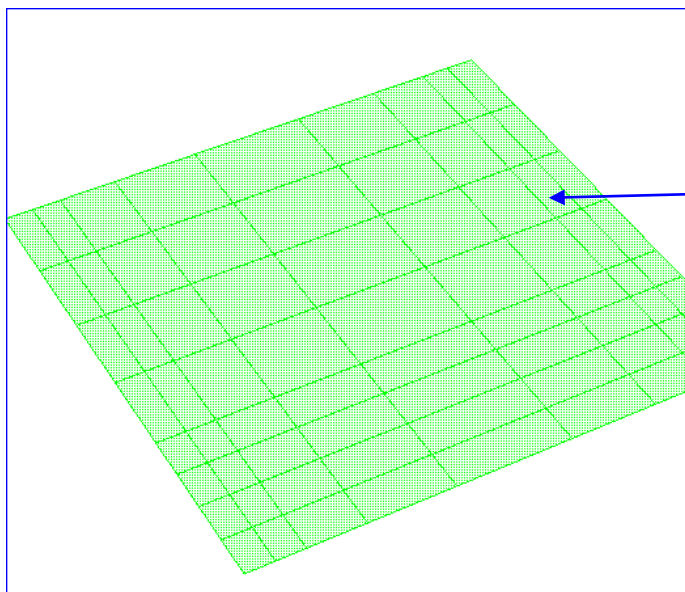


You may also benefit from limiting, or adding, objects to see in the graphic window. There are powerful features for doing so and they are available by selecting an object(s) or named set(s), **RMB** and *Visible Model*. You can also use the short commands, typically **Alt+S** (show selected only), **ALT+Plus** (add selection), **ALT+Minus** (remove selection) and **ALT+A** (show all). Some examples on how to do this is shown on the next pages.

In the example below the graphic window contains a display of a guide plane and it is shown how to add and remove the guide points Point1 -> Point5 in the graphic view. Notice that these operations are not the same as insert or delete.

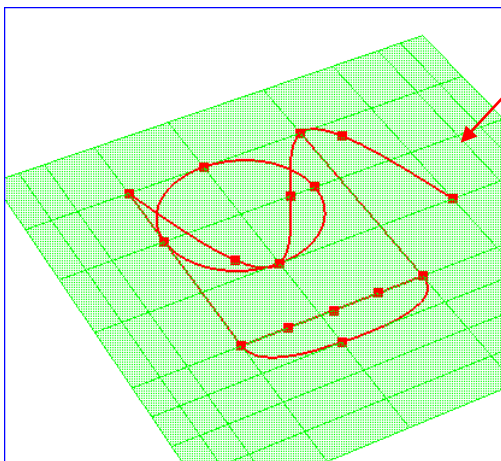
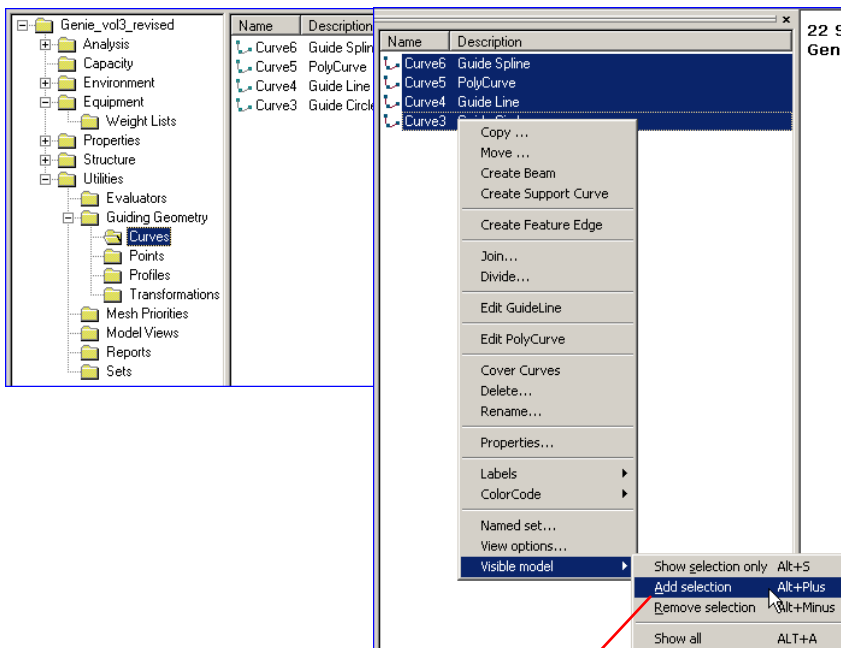


To make the points visual,
select *Add Selection* in the
Visible Model



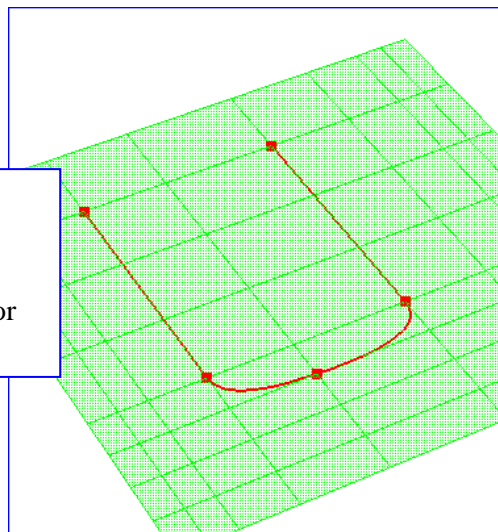
To remove the visual sight of
the points, select *Remove
Selection* in the *Visible Model*

In the example below the graphic window contains a display of a guide plane and it is shown how to add the guide curves Curve3 -> Curve6 in the graphic view. It is also shown how to visualise one of these.



On the picture to the left all above selected curves are shown.

The picture below shows the one curve only together with the guide plane. The text box explains how to do it.



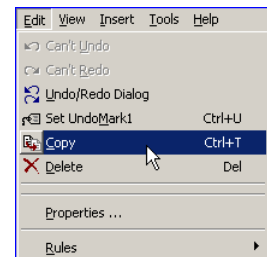
To see only one or multiple guidelines, curves or point. Just select the lines and curves you want to see, **RMB** and **ALT+S** or *Show Selection only*.

3.2.12 Delete, Move and copy guiding geometry

All these operations are done by selecting the object (in this section guiding geometries), **RMB** and select *Delete*, *Copy* or *Move*. Alternatively you can select the object and use the pulldown menu as shown to the right. It is also possible to use short commands like **CTRL+T** (same as copy) and **DEL** (for delete).

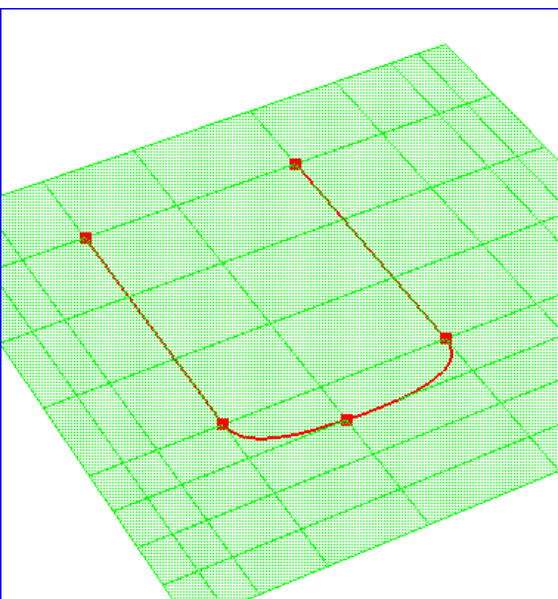
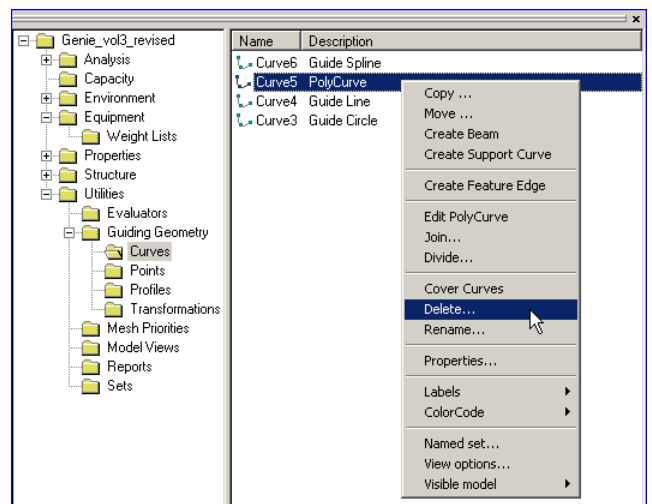
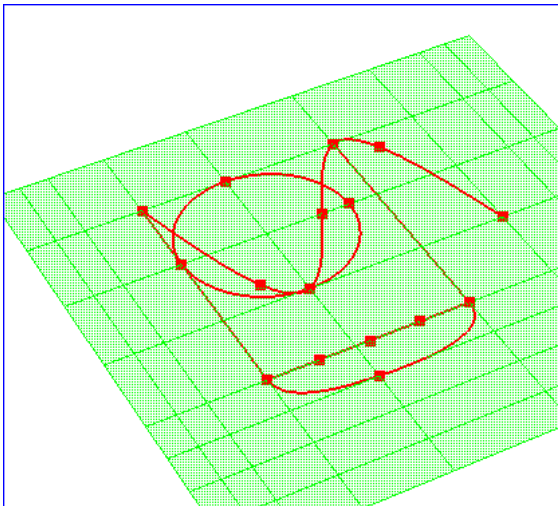
Once you have performed a copy, move or delete operation they are permanent. You may use the undo features (**CTRL+Z** for undo and **CTRL+Y** for redo) to step back or forward. Alternatively you can use the undo/redo dialog and directly go several steps back or forward; the undo/redo history is closed when saving the workspace.

In the following there are examples on how to do delete, copy translation, copy rotation, copy mirroring, copy 3 point positioning and copy using a scale factors. Move is identical to copy except that no new objects are created; the objects subjected to move are modified.



Delete objects

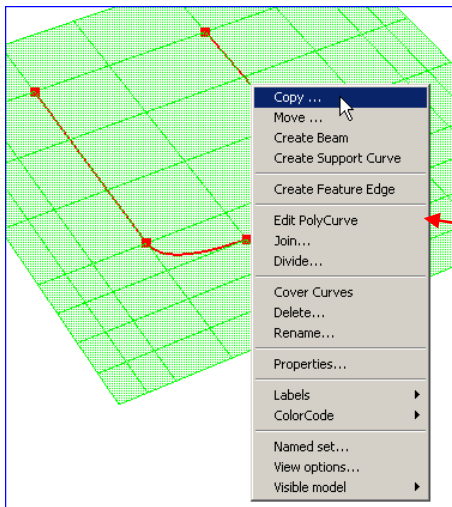
The example below shows how to delete guiding curves using the context sensitive menu (**RMB**).



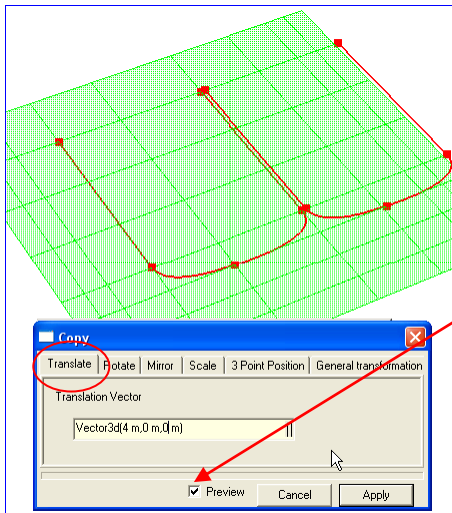
You can select several objects to delete them in one go or you can do it one by one. In the example to the left all except one curve has been deleted.

Copy/move objects using translation

The steps below show how to copy an object based on a translation vector. The move operation works the same way.



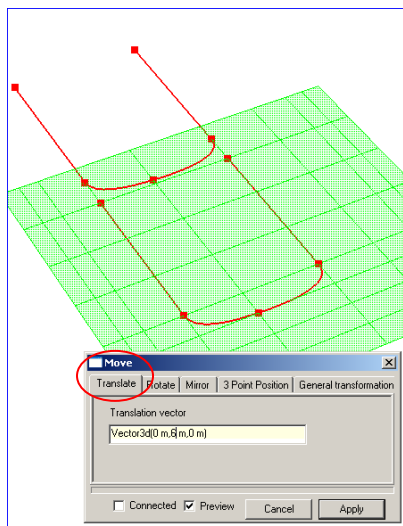
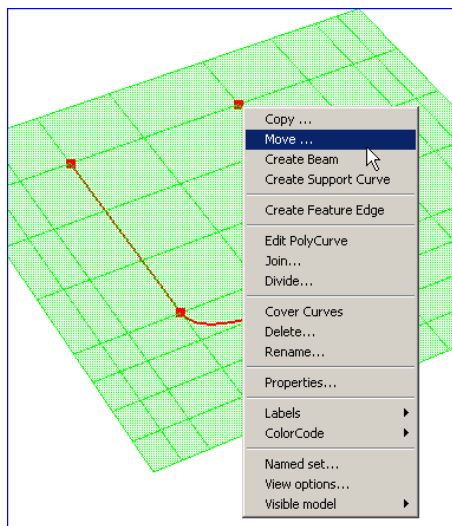
Select the guide curve to copy and then press **RMB**, select *Copy*.
For move you select *Move*.



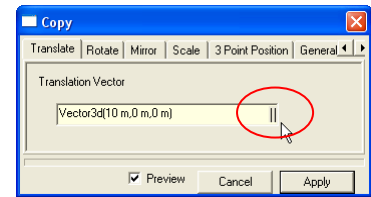
Then you type in the translation vector manually or find it from clicking between two points. If you use the *Preview* option you will see the new position before you actually perform the operation.

Automatic preview is set from the *View/Options/General*.

An example using move operation:



If you want extended control of the moving/copying you can press the symbol with the two parallel lines to the right in the move/copy dialog.



This will open the dialog shown to the right. The dialog is set up so that you can move your structure a specified length in the direction that you specify in the dialog.

The concepts are explained below:

Global c/s

Global coordinate system. Use this if you want to move your structure a specified length along an axis of the global coordinate system.

Local c/s

Local coordinate system. Use this if you want to move your structure a specified length along an axis of the local coordinate system of the structure specified in the dialog. This is useful if you want to move a straight beam along the beam's own x-axis.

Angle wrt x-axis on

Use this if you want to move your structure a specified length along an axis which forms an angle with the x-axis on one of the planes XY or ZY. The angle can be specified by typing in the degrees in the dialog.

Two points

Specify two points that give the direction you want to move. If you want you can check the "Unit vector" checkbox to use a vector of length 1.

Normal vector of the...

Guide-plane

Returns the unit normal vector of the selected guide plane.

Flat plate

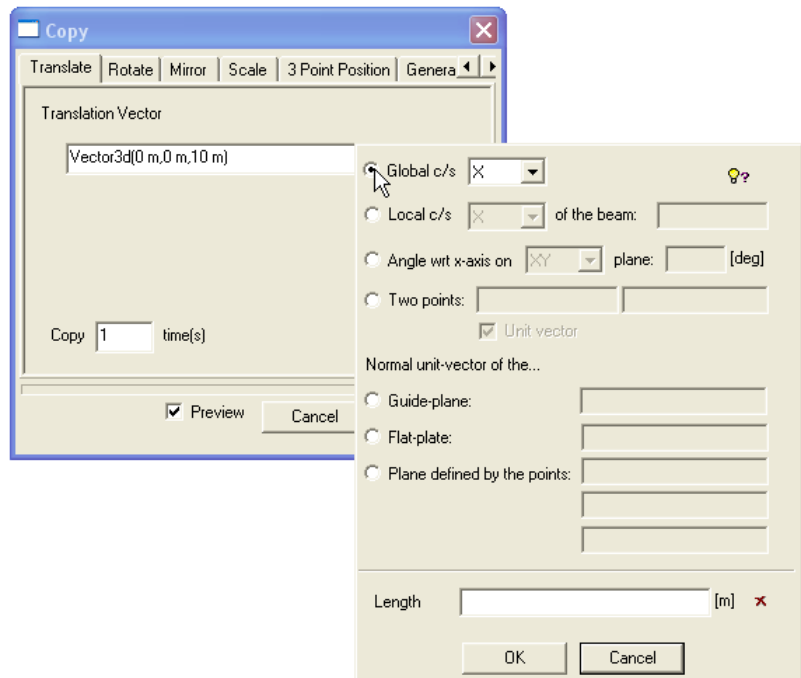
Returns the unit normal vector of the selected flat plate.

Plane defined by the points

Returns the unit normal vector of the plane formed by the three selected points.

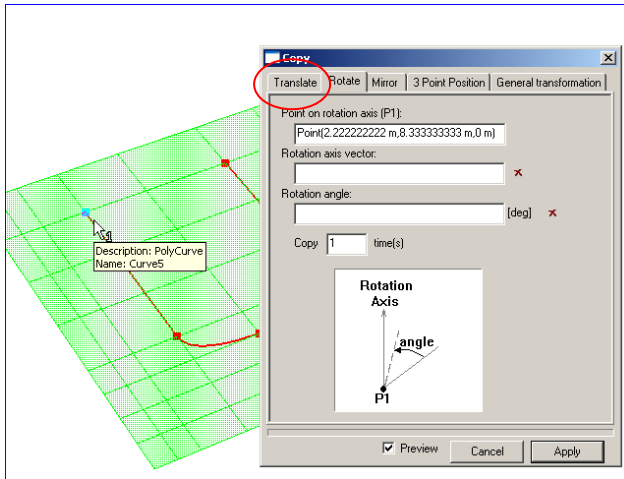
Length

Here you type in the distance you want to move the structure.

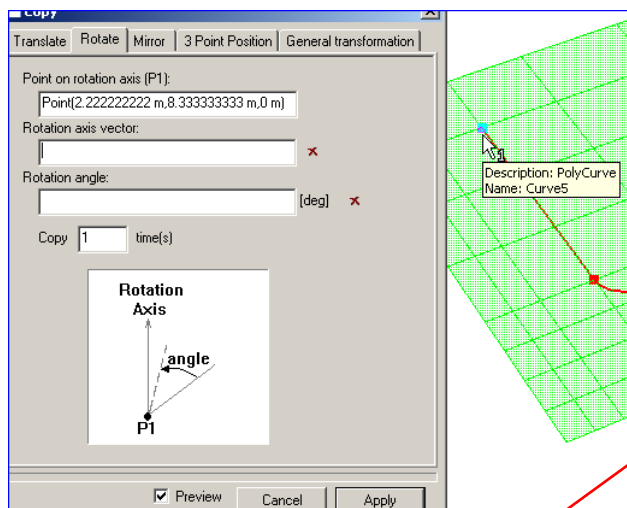


Copy/move objects using rotation

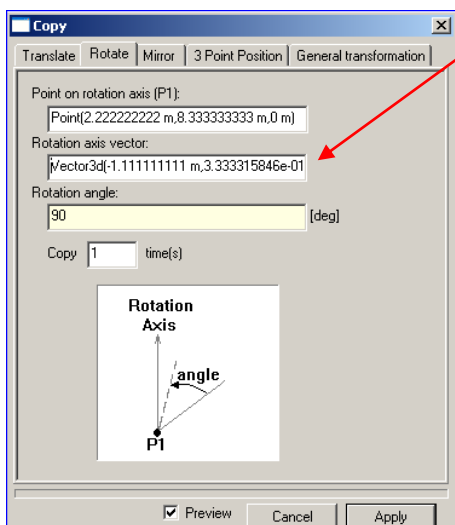
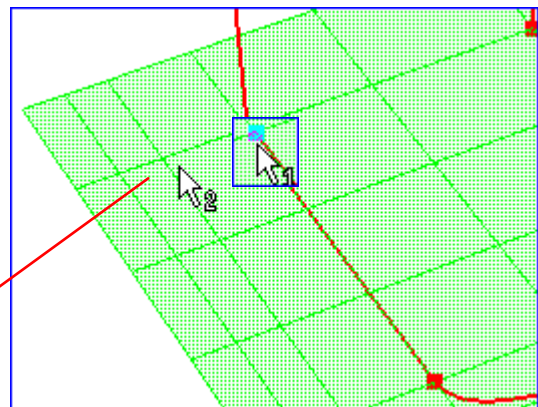
A copy rotation operation requires input on rotation point, rotation vector and rotation angle.



The first task is to define a rotation point either from manual input or by picking a snap point from existing objects. In this case the rotation point has been selected from the graphical window.

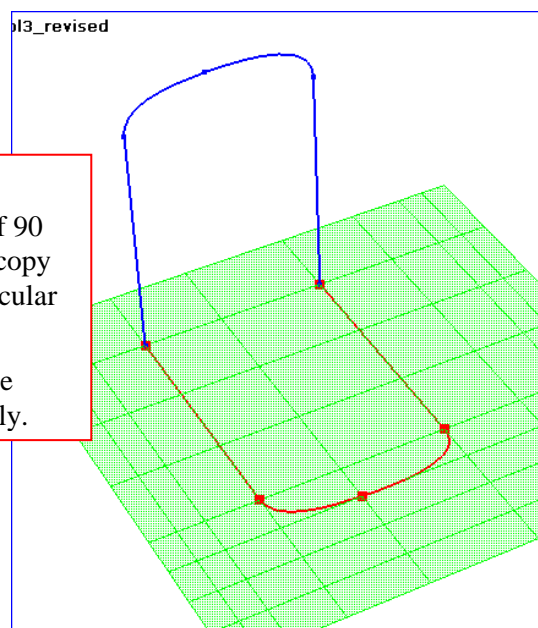


Secondly, you specify the rotation axis vector either by manual input or by clicking between two snap-points (like shown below).



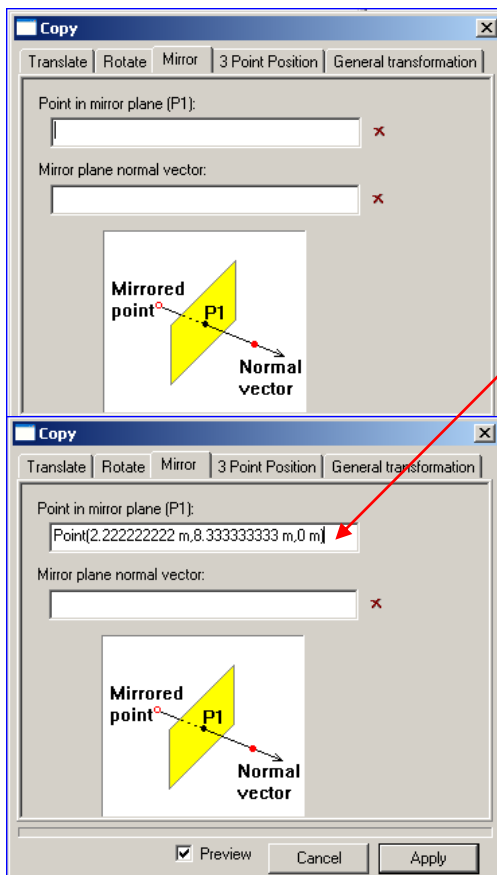
When using a rotational angle of 90 degrees, the new copy (blue) is perpendicular to the old one.

Remember that the right and rule apply.

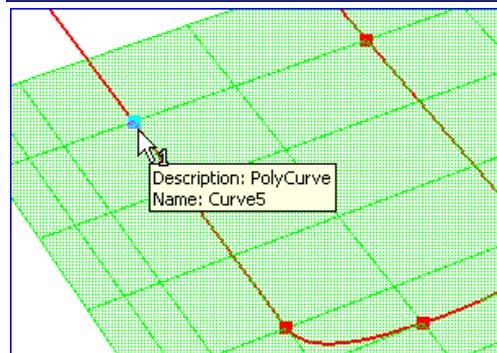
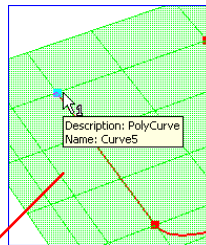


Copy/move objects using mirror

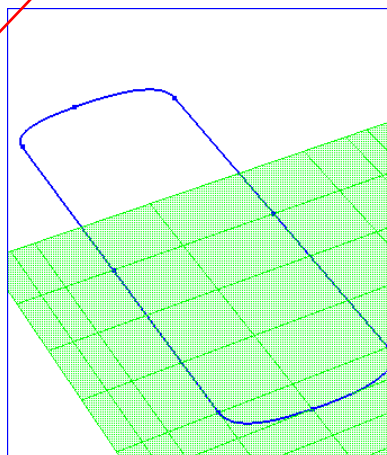
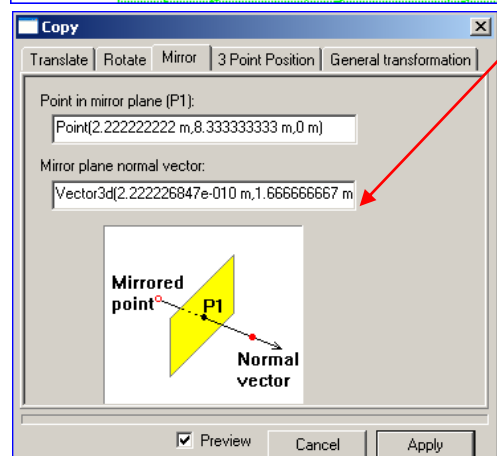
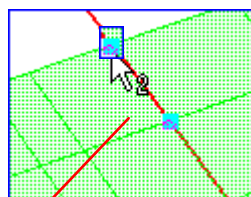
A mirror operation requires two input parameters; a point in the mirror plane and a normal vector to the mirror plane.



The first task is to specify a point in the mirror plane (in this case from a snap point).



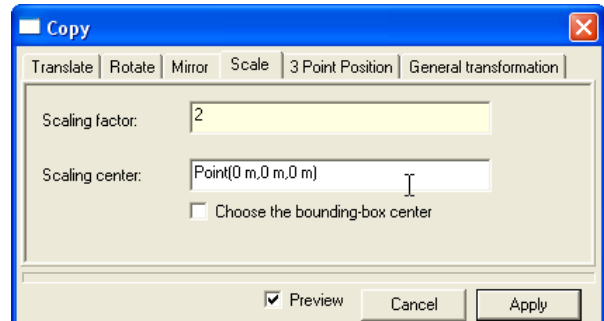
Second step is to define a vector normal to the mirror plane, in this case picked from the graphical window.



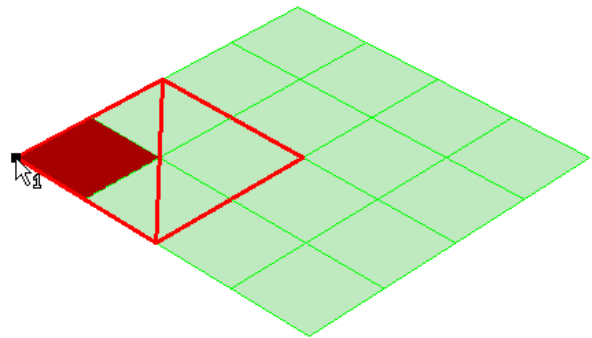
Copy/move objects using scale

A scale operation requires two input parameters; the scaling factor and the point to be used as the scaling center. The scaling factor should be set to a value above 1 if you want to increase the size of your selection and below 1 if you want to decrease it.

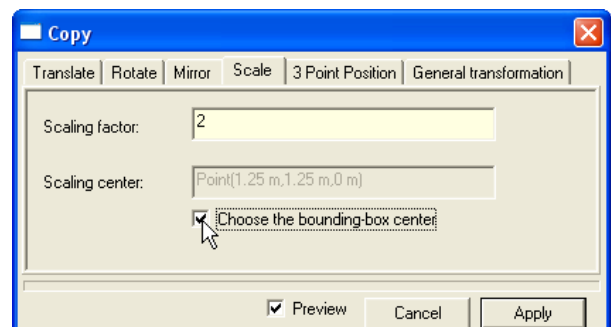
You can select a scaling center by clicking in the graphics or by typing in the coordinates in the dialog.



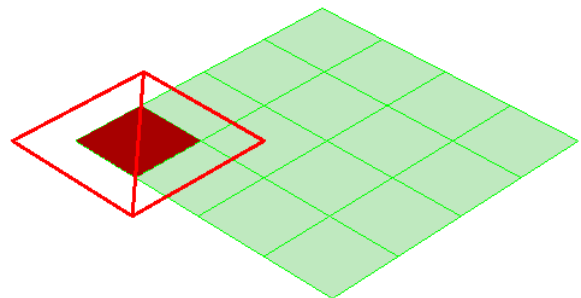
The preview shows you what the result will look like.



By checking the “Choose the bounding-box center” checkbox you can use the center of your selection’s bounding box as the scaling center.

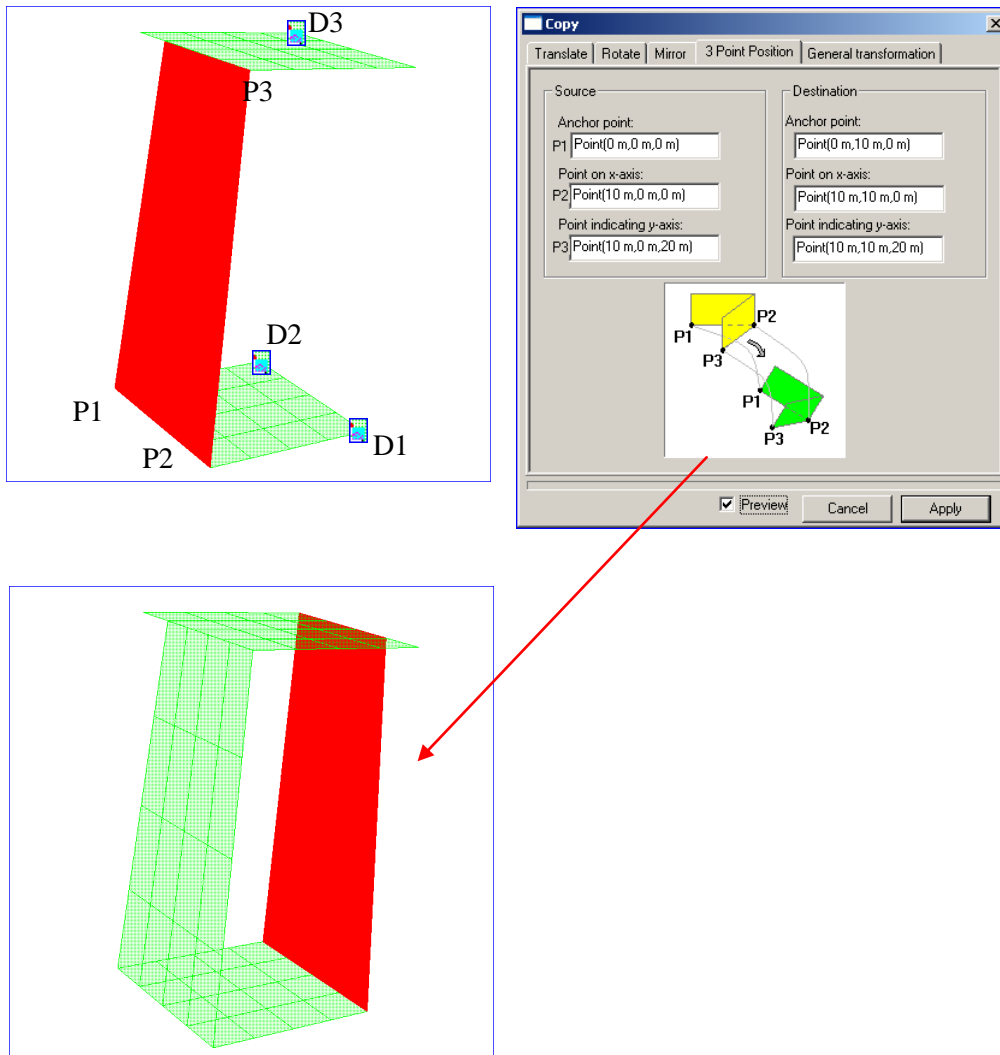


The preview shows you what the result will look like.



Copy/move objects using 3 point position

A 3 point position copy/move operation requires 6 input values; it is necessary to specify 3 reference points and 3 target points. This is illustrated by copying the vertical, yet skew, guide plane from one side to the other. The source points in this case are denoted P1, P2 and P3 while the destination points are named D1, D2 and D3 on the picture below. This copy/move technique is particularly of advantage when modelling battered jackets.

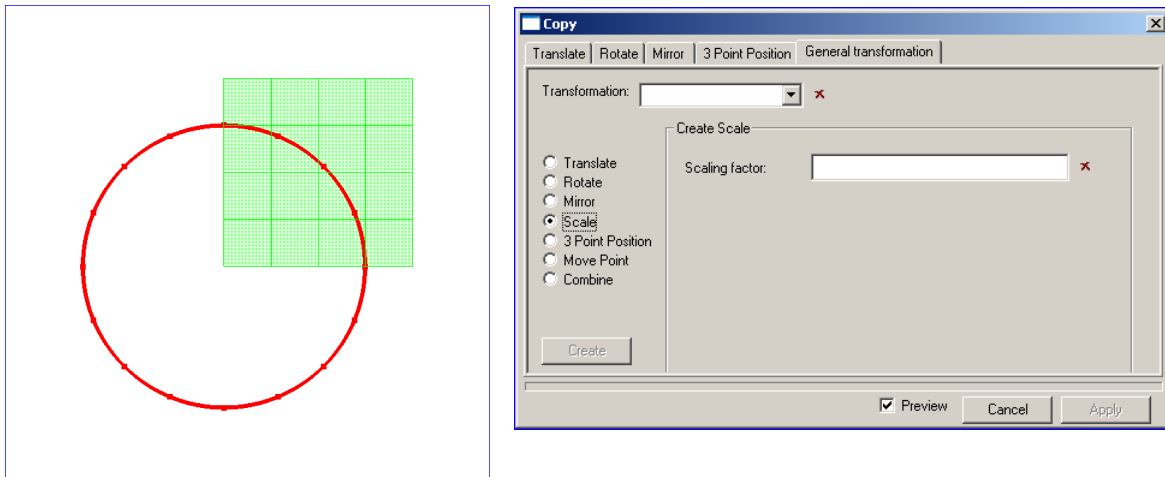


Copy/move objects using 3 scale

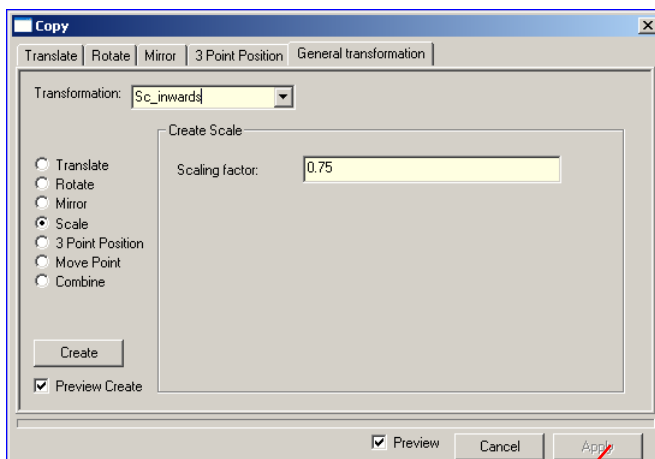
The scale function is available from the General Properties tab. In general all copy/move operations as explained above can be described as General Transformations whereby a specific and named transformation can be repeated as necessary. Typically, you can specify a copy/move translate transformation with a constant copy vector (2m, 0m, 0m). When referring to this transformation, all objects will be copied or moved 2m in x-direction.

The scale function is particularly useful when you want to extend the size of a new object relative to the existing. Notice that the scale function is relative to the origin (0, 0, 0). An example of its utility is when a constant distance between two curved lines is desired. An example is given below.

First step is to make the scaling transformation. Select the objects to be scaled, then **RMB** and *Copy* or *Move*. Under *General Transformations* the option *Scale* should be chosen.

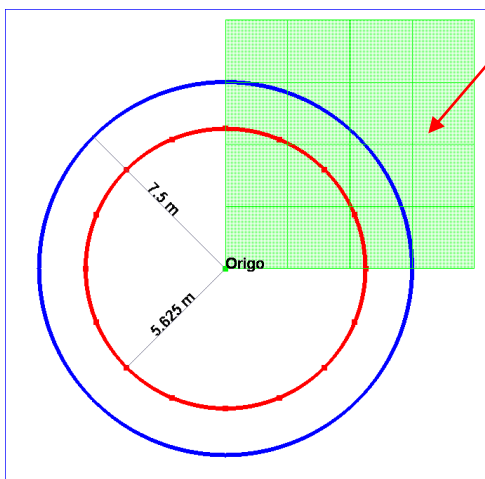


Define a name to the transformation (here Sc_inwards) and add a scaling factor (here 0.75).



The transformation is defined when clicking on the Create tab, if you click on Preview Create you will see the effect of the transformation.

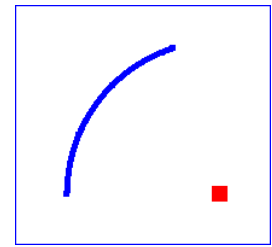
Click Apply to make new objects using the Sc_inwards scaling transformation. The scaling transformation can also be used at a later stage.



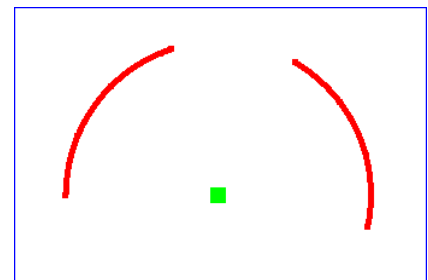
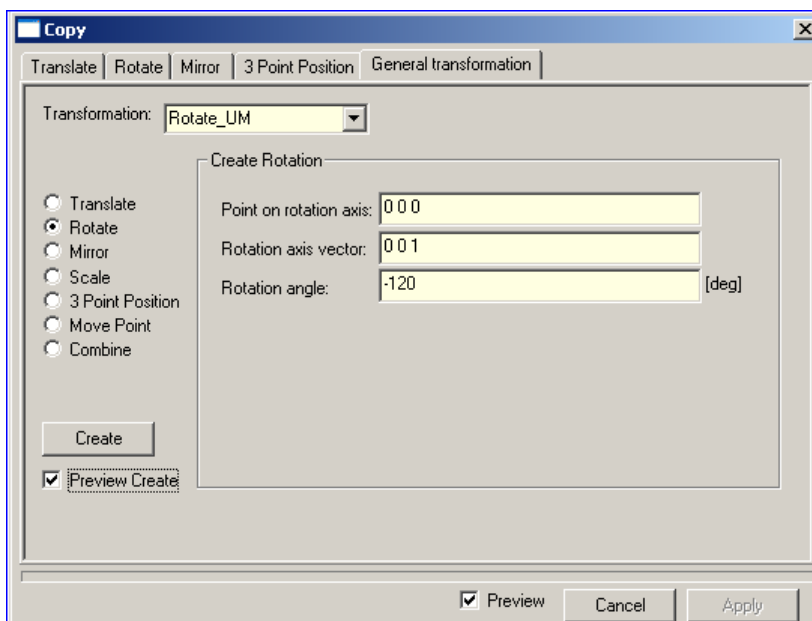
New guide curves are now made with the radius is 0.75 times the length of the original radius. In this case the origin is in the middle of the circle, and transformations are relative to this point. Hence, two circles with different constant radius are made.

The scaling is relative to the origin. The radius of the reference curve is 7.5m meaning that the radius of the resulting curve is $7.5\text{m} \times 0.75 = 5.625\text{m}$.

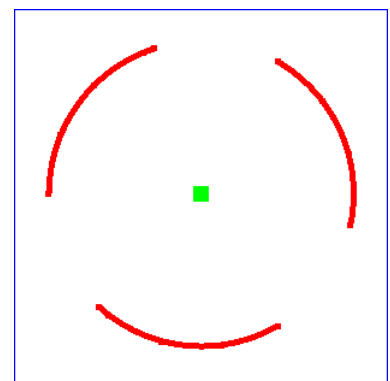
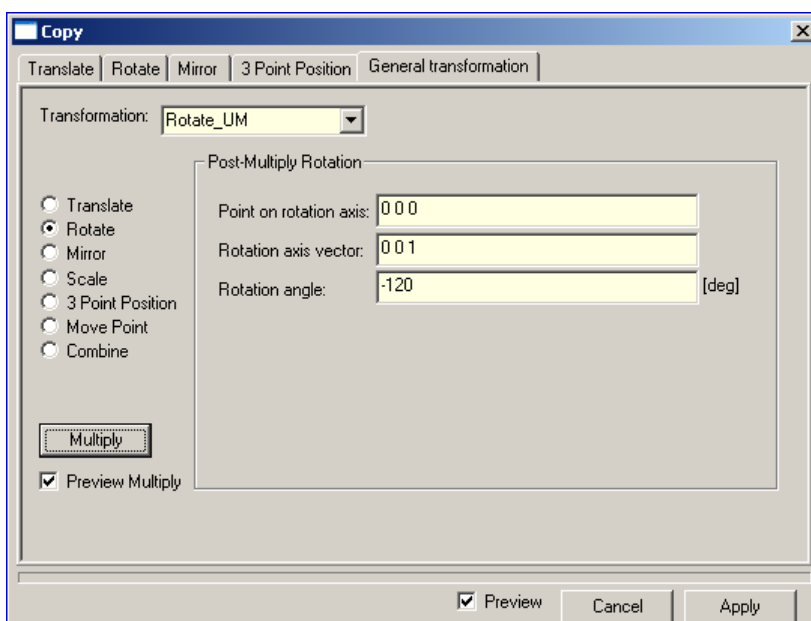
Another scenario of using the general transformation can be when you have a combination of rotation and scaling. A circle segment is used to illustrate how this can be done (the rotation is around the high-lighted point).



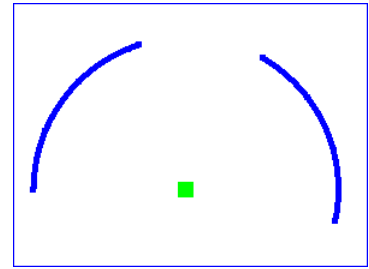
First step is to make a *Transformation* and name it to “Rotate_UM”. Select the circle segment, **RMB**, *Copy* and click the tab *General Transformation*. The effect of the transformation can be seen since the “Preview Create” is checked.



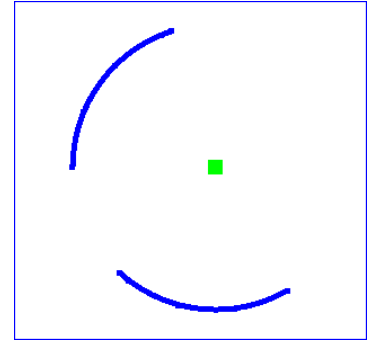
The transformation has not yet been created, to do this press the “Create” button. The dialogue box now changes as shown below; multiply means in this context double the rotation angle.



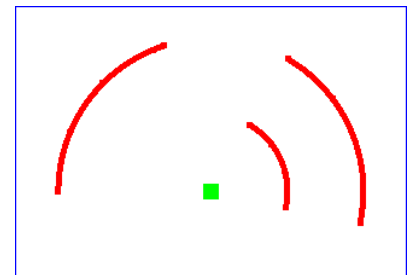
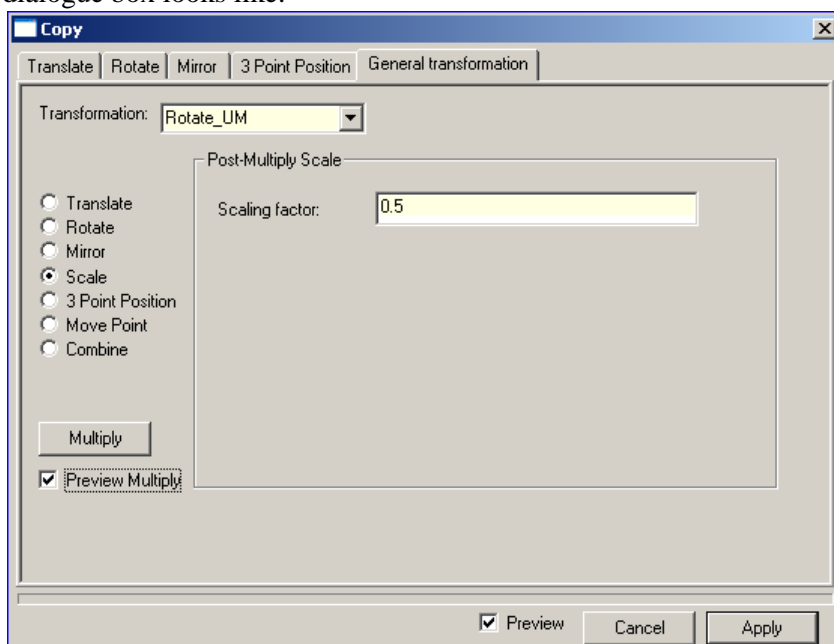
To make one new circle segment (copy the initial segment 120 degrees) click “Apply”.



To make a new circle segment by using the multiply feature you need to click “Multiply” prior to “Apply”.

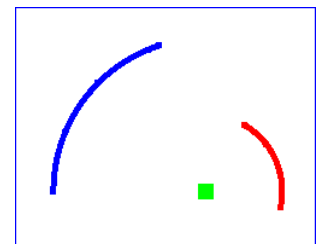


The general transformation “Rotate_UM” can now be multiplied with another transformation like e.g. scaling down to half size (factor of 0.5). When selecting scaling as the post transformation action, the dialogue box looks like:



Click “Multiply” to modify the transformation “Rotate_UM” and the “Apply” to rotate and scale the initial segment in one operation. The picture to the right shows the new segment (in red colour). The commands generated are shown below.

```
Rotate_UM = Rotate(Point(0, 0, 0), Vector3d(0, 0, 1), -120);  
Rotate_UM.scale(0.5);
```

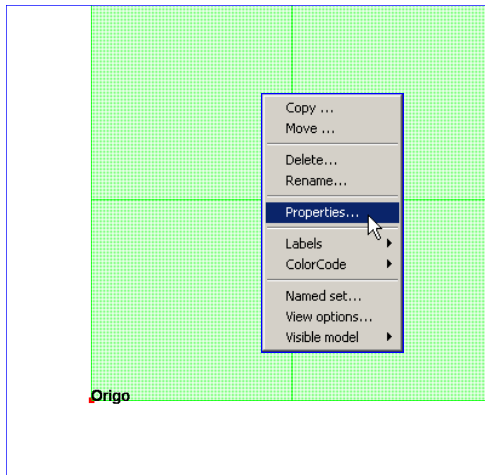


3.2.13 Modify guiding geometry

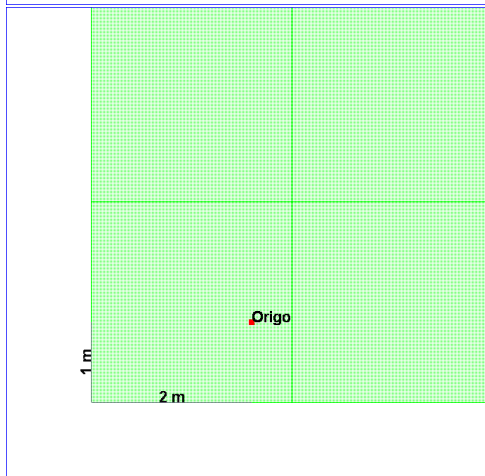
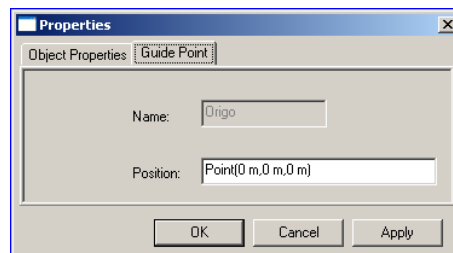
In general all objects can be modified after they have been created. Typically this is done by selecting an object(s) from the graphic window or from the browser (see previous Section *Find, select and display guiding geometry*), **RMB** and *Properties, Edit, Join or Divide*. This Section gives some examples on how to do these operations.

The most commonly used operations are the *Join* and *Divide*. Typically when building up complex curves to be used when creating structure or splitting structure or when dividing curves defined by importing data from other sources.

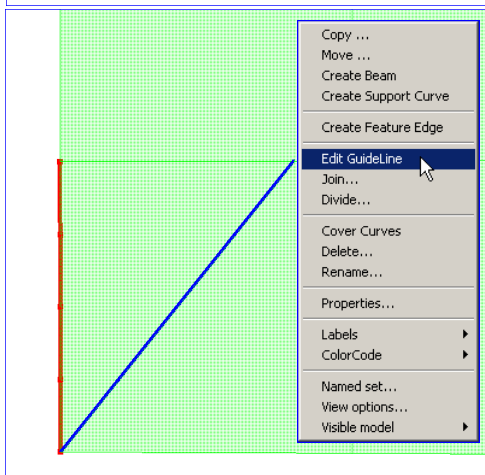
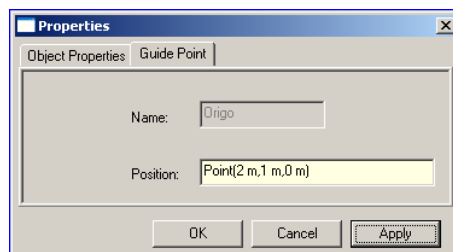
Modify using Properties or Edit



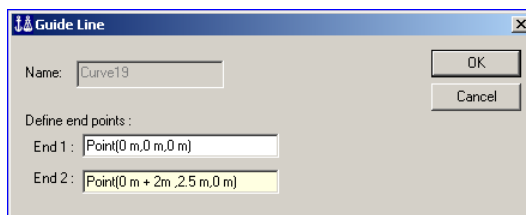
Select the guide point *Origo*, **RMB** and *Properties*.



Modify the new position to (2m, 1m, 0m); the guide point is automatically moved to its new position. The same result can also be achieved by using the *Move* command.



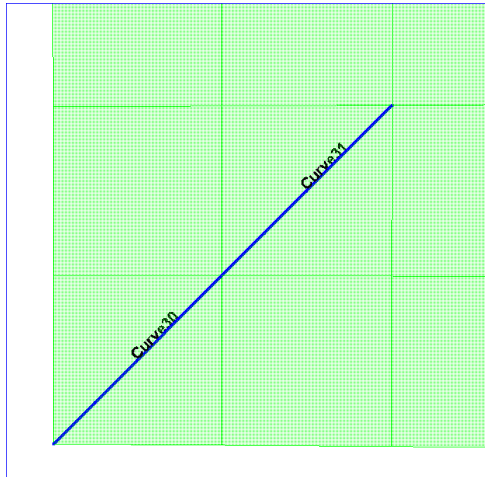
The same principles apply when using the edit function. Select the highlighted line, **RMB** and *Edit*. In this example, the x-coordinate at end point 2 has been edited ($0m + 2m$) and the line is automatically modified (the blue line).



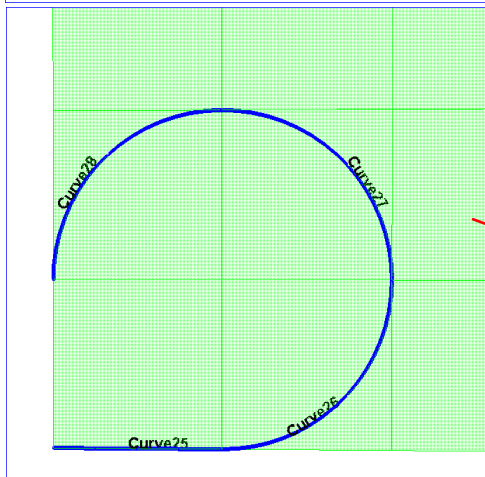
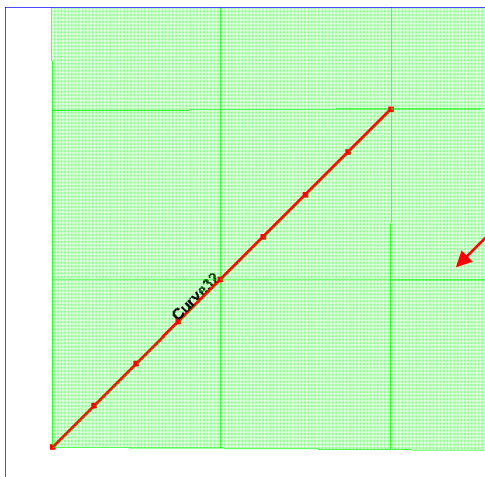
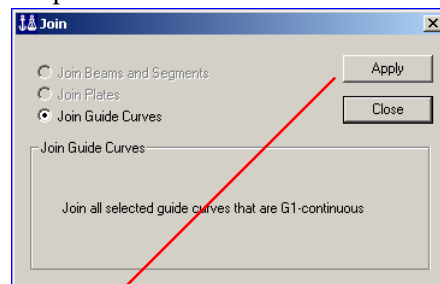
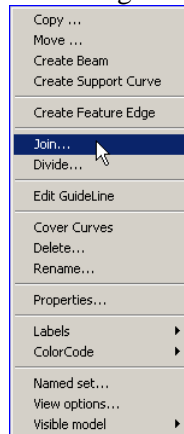
Modify using Join

When joining guiding lines (straight lines or curves) it is required that they are continuous (G1 continuous) at the connection points, i.e. the tangent of the line must be the same for both lines to be connected. This means that lines can not be joined to form what is known as poly-lines. Lines that are joined become a composite curve.

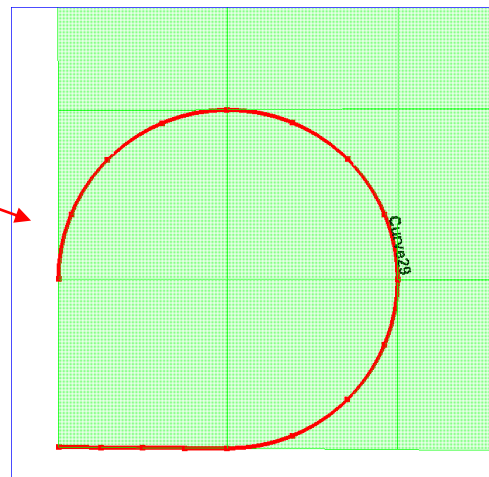
Three examples are given to illustrate the *Join* command.



Select the straight lines *Curve30* and *Curve31*, **RMB** and *Join*. This will give the composite curve *Curve32*.

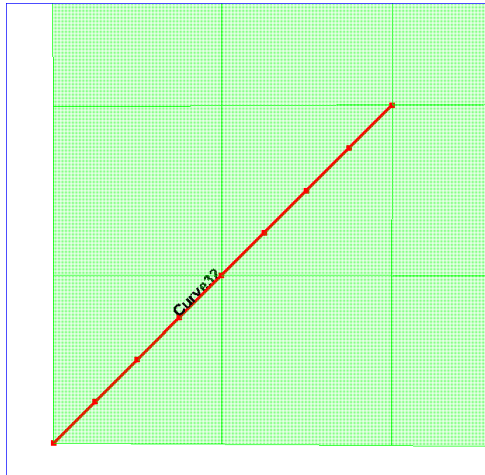


The straight curve *Curve25* and the guide arc elliptic curves (*Curve26* -> *Curve28*) becomes the composite curve *Curve29*.

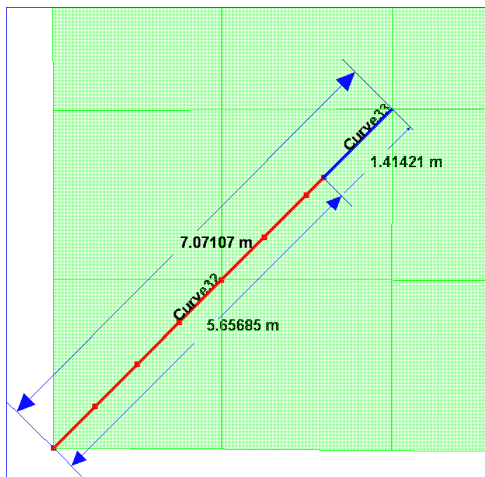
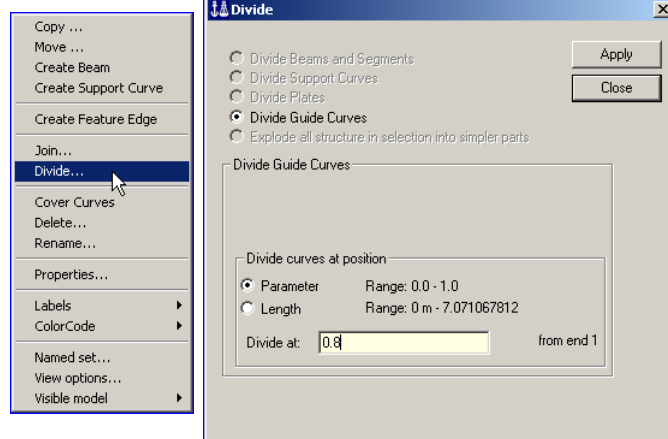


Modify using Divide

Divide curves can be done using a parameter (relative length) or true length. In both cases the division is relative to the start position of the line (end 1). Four examples are displayed to show how the *Divide* function can be used.



Select *Curve32*, **RMB** and *Divide*. Use parameter 0.8 to divide the curve.

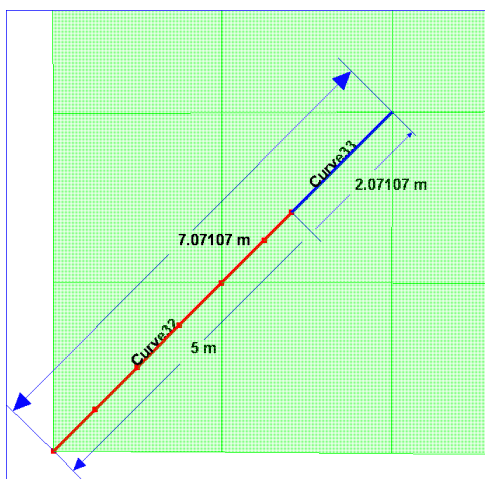


Curve32 is split in two; *Curve32* and *Curve33* in this case.

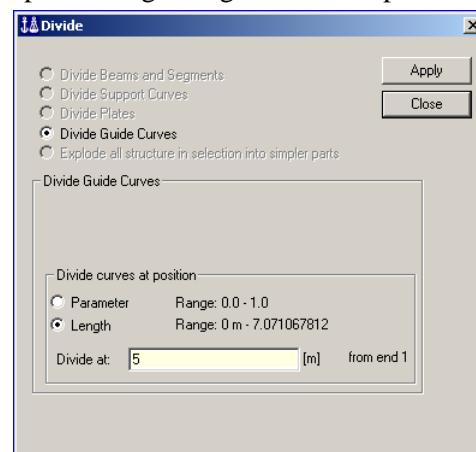
The length of the modified *Curve32* is
 $7.07107 \text{ m} \times 0.8 = 5.65685 \text{ m}$.

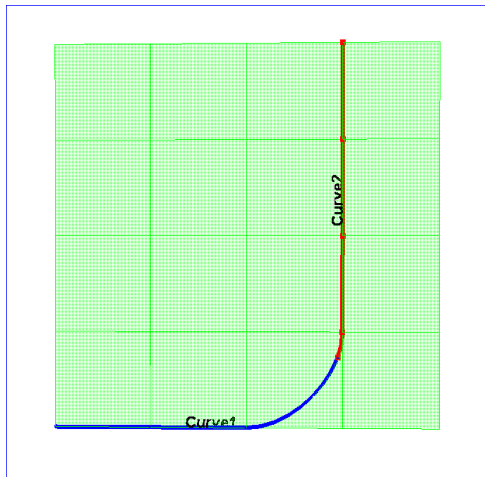
Similarly, the length of the new *Curve33* is
 $7.07107 \text{ m} \times 0.2 = 1.41421 \text{ m}$

Notice also that the dialogue box shows the total length of the line (in the length option).

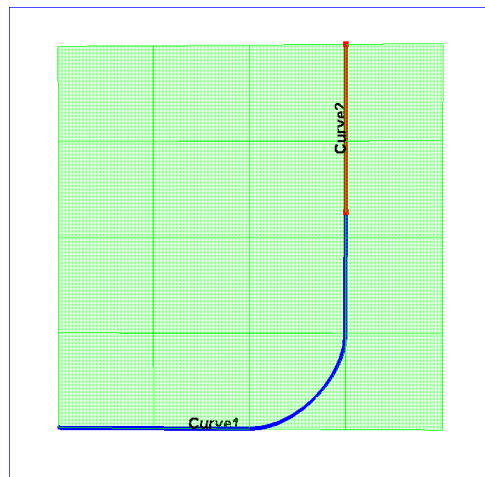
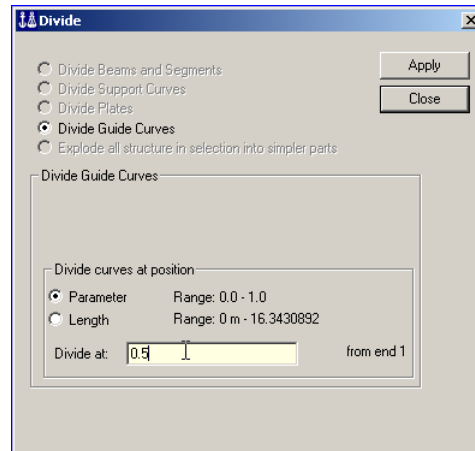


The picture to the right shows *Curve32* split with the length option using a length of 5m to split the line.

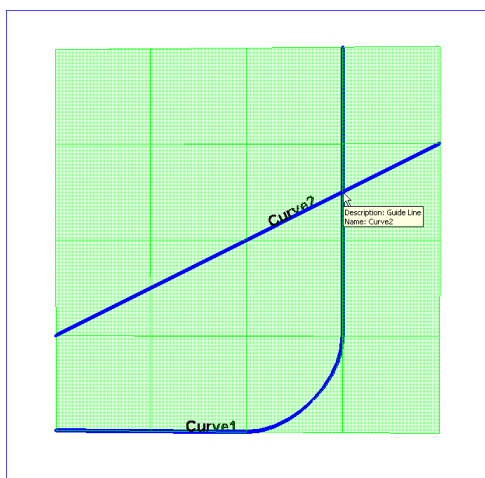
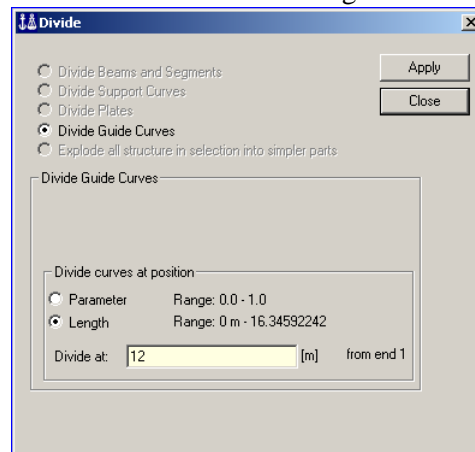




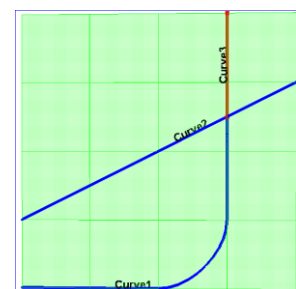
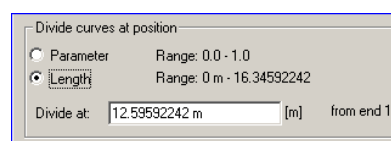
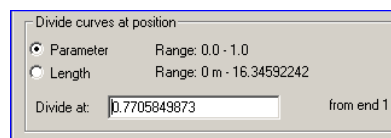
In this case the poly curve *Curve1* is divided using parameter 0.5 and split into a modified *Curve1* and a new *Curve2*.



The poly curve *Curve1* is divided using a length factor 12m. Notice also that the total length of *Curve1* is shown.



In the previous examples, the division factors have been specified manually. It is also possible to divide using the positions of snap points (vertices or connection points). In the example to the left the intersection point between *Curve1* and *Curve2* is used to divide *Curve1*.



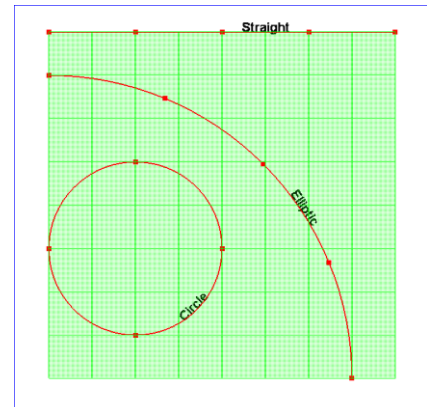
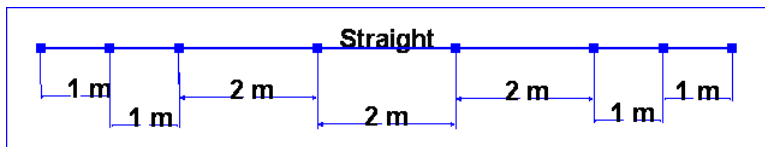
3.2.14 Snap points

Snap points may be used during modelling tasks to find co-ordinate values, vectors and so on from the graphical window. Typically, when inserting a beam you can click on two snap points and a beam is inserted in-between. Similarly, you can find a vector by clicking between two points.

Snap points are automatically made when creating guiding geometry, where guide curves intersect and where structural parts intersect (typically two beams crossing each other). Some examples on snap points belonging to guiding geometry are shown below.

For guide curves of type straight and elliptic there are 5 points to describe 4 primitive parts with equal lengths. Similarly for circular arcs there are 4 points to describe 4 primitive parts with equal lengths.

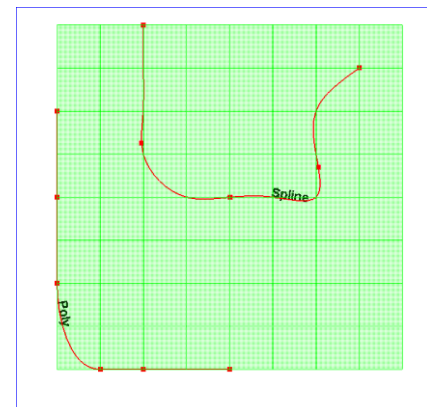
For these guide curves you can modify number of snap points and the relative length of each primitive from the command window or on the journal file by using the script command. Typically for the line Straight you can modify by `Straight.spacings(Array(1,1,2,2,2,1,1));`



For composite curves the number of snap points depends on how many points you used to describe the curve. The number of snap points for a spline depends on how many are needed to define a spline (i.e. how many primitive parts are necessary)

In the example to the right, 6 points were used to make the poly curve; hence 6 snap points. Similarly, 11 points were used to make the spline but only 5 points are needed to make the spline; hence 5 snap points.

Snap points on a composite curve depends on the numbers of snap point on curves to be joined.

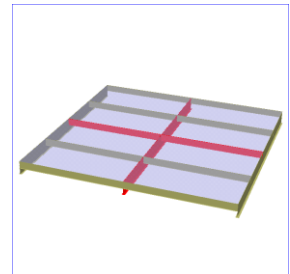
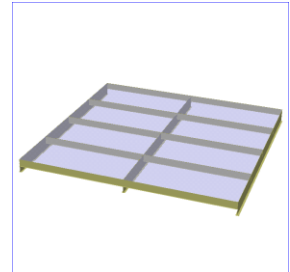


3.3 Create a structure concept model

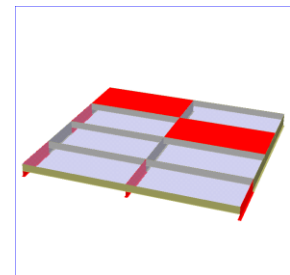
This Section will show you how to make a structure concept model build up of plates and beams. A plate can be in a plane or have complex curved geometry – it may also contain stiffeners. The structure may be built up using a bottom-up modelling technique (make small parts of the structure and assemble them), a top-down modelling approach (where large parts are made and split if needed to refine) or a combination of the two approaches.

The stiffened panel can be made in two different ways. It consists of the plate and stiffeners in two directions.

When using the top-down modelling approach (where the strategy is to model as large as possible and refine when needed) there is one plate and a total of 8 stiffeners. Two of the stiffeners are highlighted and as can be seen it is continuous over intersections with other stiffeners or plates.



The bottom-up modelling alternative is used to the right, and as can be seen there are more plates (8 plates) and stiffeners (22 stiffeners) needed to build the stiffened panel. Some of the plates and stiffeners are highlighted.



It is common for all structure modelling to be based on:

- Manual input (i.e. co-ordinates or points)
- Snap points in existing structure or guiding geometry,
- Cover, skin, loft, curve-net interpolation or extrude operations using guide curves or beams
- Existing structure using copy operations

All structural parts may be modified after they have been created, like for example in split operations.

The following Chapters describe basic modelling features on how to create a structure concept model for use in hydrostatic, hydrodynamic and structural finite element analyses. They also give examples on typical structures that are common to the maritime and the offshore industry.

3.3.1 Basic modelling features

Basic modelling features include

- Insert objects using manual input, skin, loft, curve-net interpolation, cover or extrude. The manual input apply to flat plates and straight beams
- Divide and punch objects using existing structure, curves or temporarily planes as reference
- Join objects that are continuous (i.e. constant tangent) in the connection area

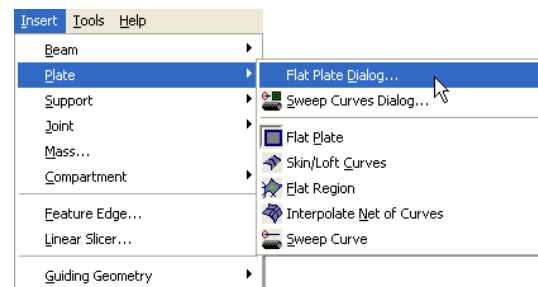
- Copy, move and edit objects (for references on how to do these, please consult the previous Chapter Create Guiding Geometry)

Examples on basic modelling are given in the following, notice that the focus is on plate modelling. Modelling beams and stiffeners is documented at the end of this Chapter.

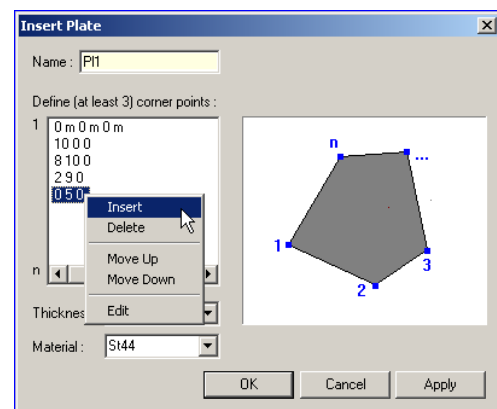
The commands for basic modelling features can be executed from the pull-down menu, the tool buttons or from the context sensitive menu.

3.3.1.1 Insert using manual input

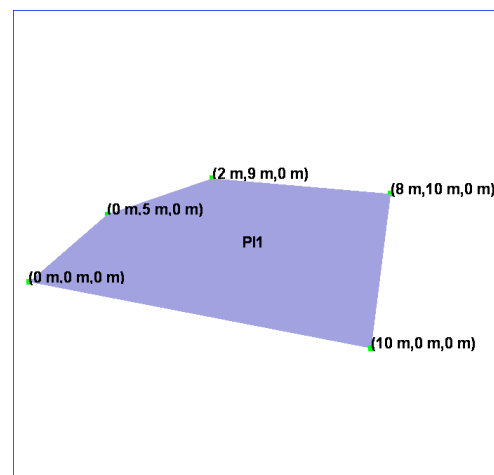
A flat plate can be inserted using the pull-down menu **Insert/Plate/Flat Plane Dialog**.



It is possible to create triangular plates, triangular plates or plates with multiple edges. This example shows how to create a plate with 5 edges. In the dialog box for flat plates you can insert the necessary number of points by inserting new rows, delete them or move them up and down. To edit or modify the co-ordinates of a row, double-click the row. The co-ordinate values may be typed in manually or found from the graphics window.



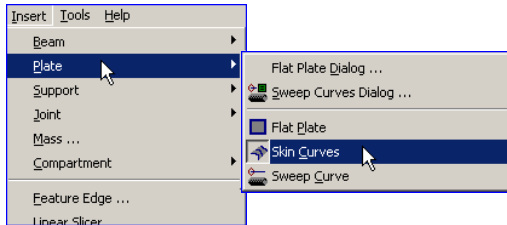
The plate *P11* is now created with 5 edges with the co-ordinate values as specified in the input dialog box.



3.3.1.2 Insert using skin

There are two ways of doing skin; between two guide curves at a time or between several curves at the same time. The result may be different depending on the curve types.

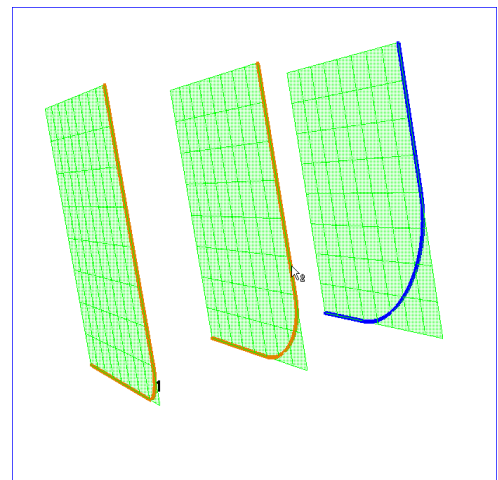
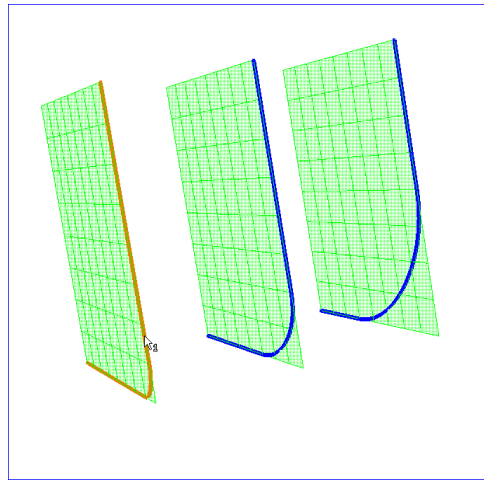
Skinning curves can be accessed from **Insert/Plate/Skin Curves** or from the tool button.



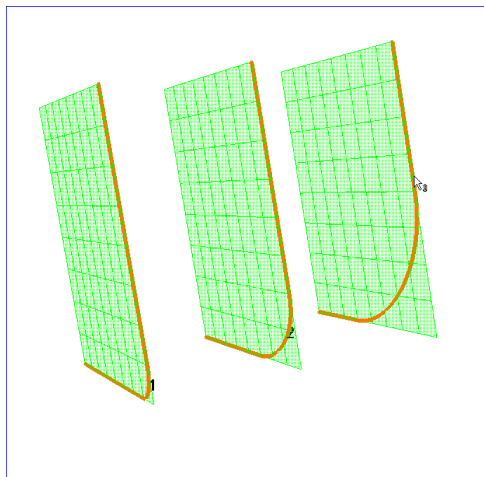
When using the skin operation you need to select (click on) which curves to skin in-between. To stop the input sequence you double click the curve. Typically, between three curves it will look like:

When moving the mouse over the curve it becomes orange.

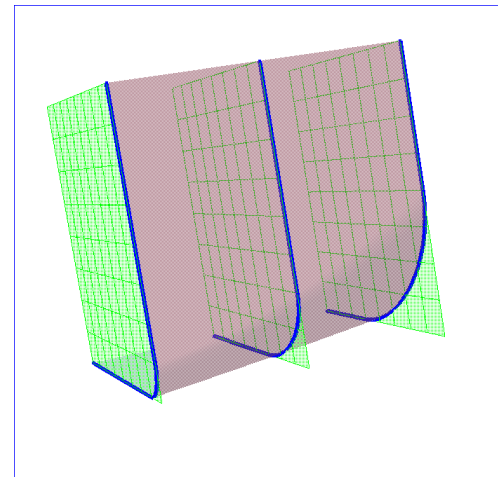
A selected curve stays orange and is denoted the sequence number. To the far right you can see that the first curve in the input sequence is denoted "1".



The picture to the right shows that two lines have been added ("1" and "2") and the third is due to be selected.

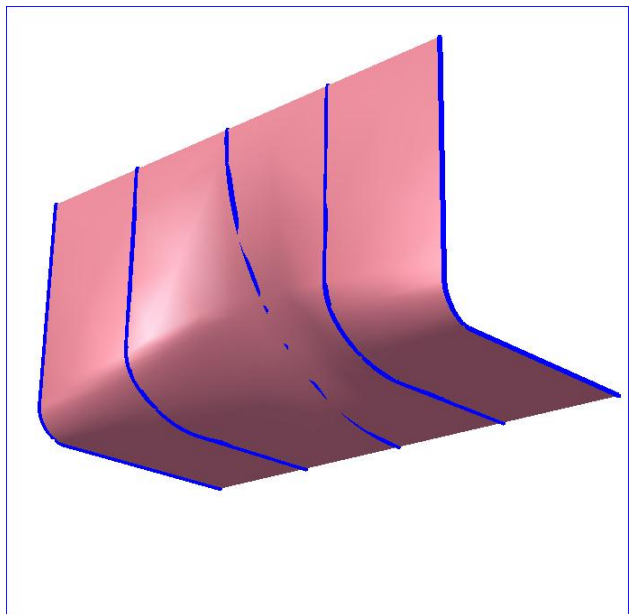
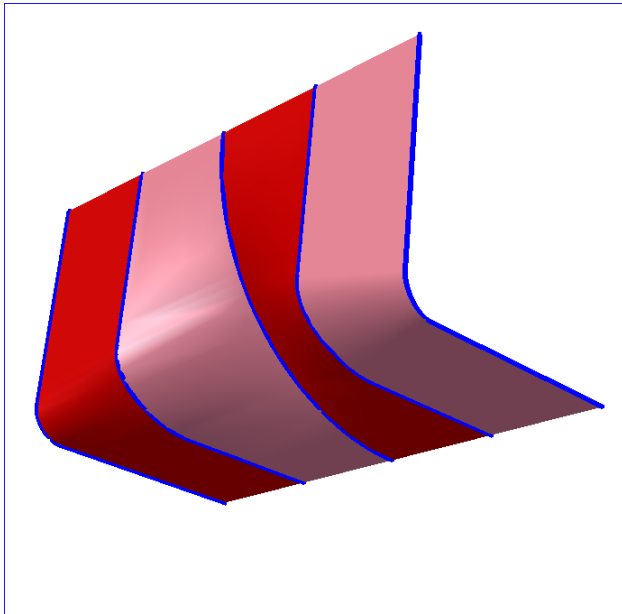


The picture to the far right shows the plate as a result of these operations – notice that the third curve is the stop position and it is necessary to click on this curve two times.



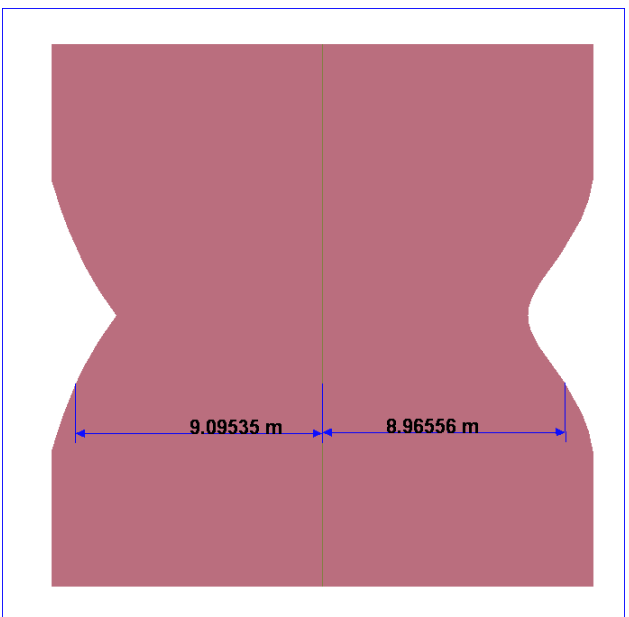
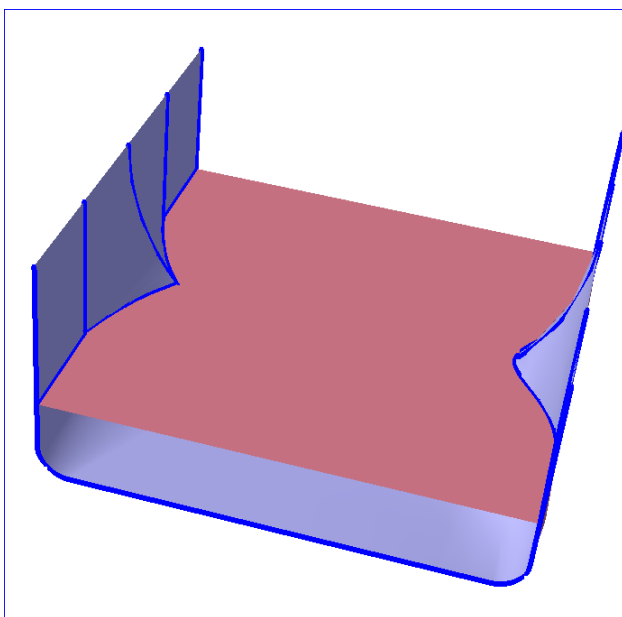
When making plates based on skinning between curves, there will always be full connectivity of the plates along the curve. In other words, there will be no cracks or openings. However, the degree of smoothness depends on how many curves are selected and how rapid the changes in curve occur.

In the example below there are 5 guide curves. The picture to the left shows the 4 plates generated by skinning in between two curves at a time (some of the plates are highlighted). When skinning between all curves in one operation, the curvature of the plate (only one plate in this case) becomes smoother, see the picture to the right.



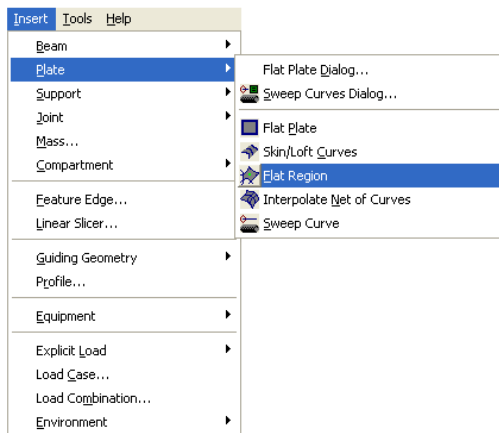
To see the difference in the two approaches a plate has been inserted and a divide operation has been performed (see later how to do this). As can be seen below there is a significant difference in the topology along at a constant elevation along the plates. On the left hand side there is no continuation in the plates around the third guideline, while the right hand side shows a continuation.

The reason for this is that skin curves between multiple curves will generate a continuous surface, while skinning between two curves at a time will ensure continuation between the two curves only, but not continuation between the other curves.



3.3.1.3 Insert using flat region

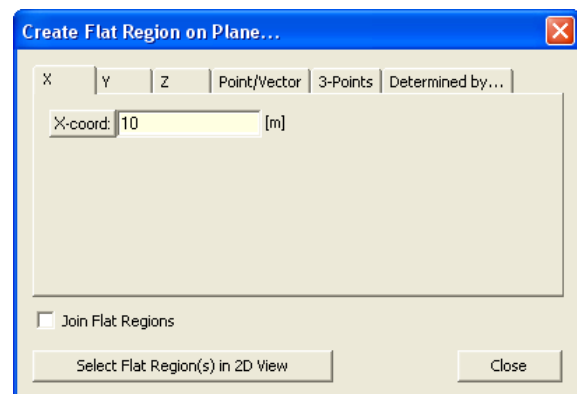
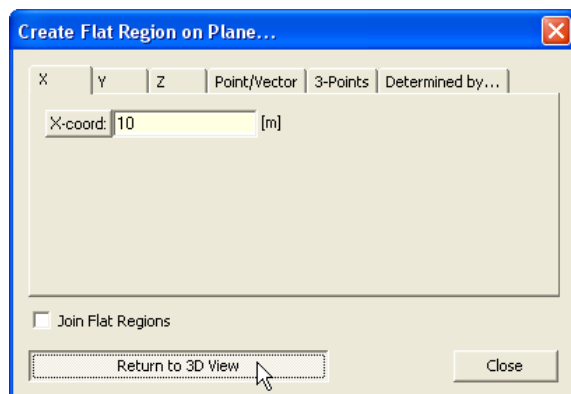
Flat region constructs a flat plate on a user-defined plane, bounded by visible guiding curves, beams or intersecting plates.



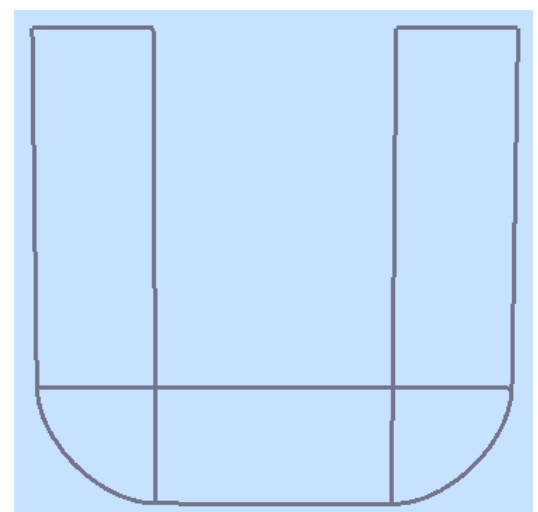
The longitudinal plates of a simplified hull will be used to explain how to create plates using “Flat Region”.

This example is chosen because using “Flat Region” is a quick and safe way to create transverse web frames.

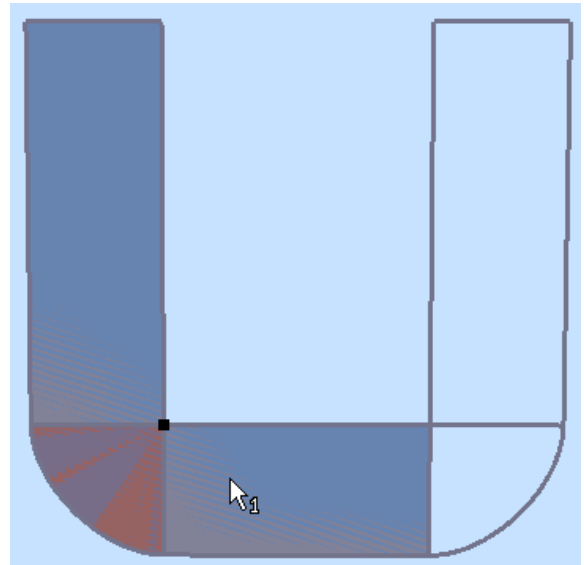
First we want to insert plates in an intersecting plane at $X = 10$ m.



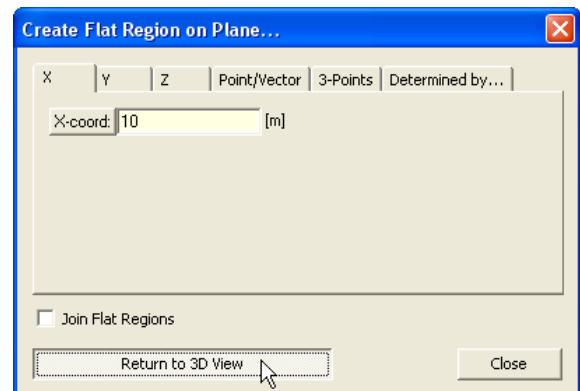
By clicking the button “Select Flat Region(s) in 2D View” the graphics window show the plane you have selected. Intersecting plates are shown as grey lines where they cross the plane. In this example there are 5 bounded areas that can be used for creating flat plates.



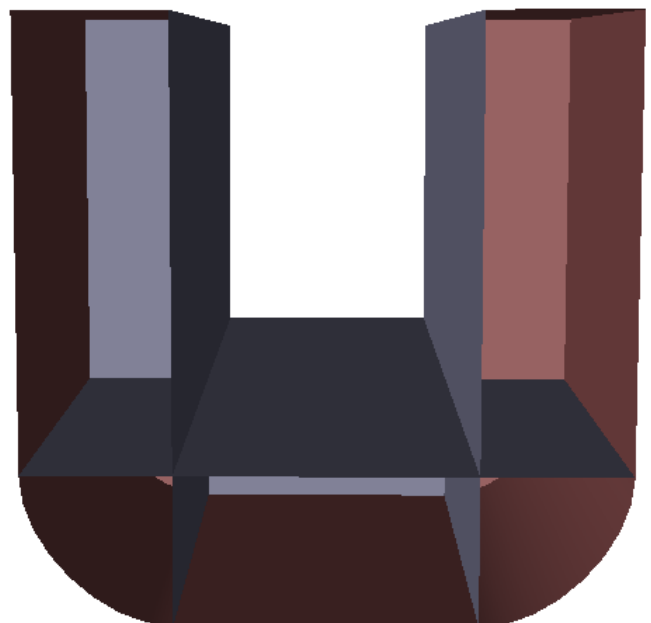
Three of the five bounded areas have been clicked. In this case, the checkbox “Join Flat Regions” has been left unchecked. This means that the plates you create are not joined as you make them. The plates’ normal vectors are not necessarily in the same direction. This can be seen on the illustration where one plate appears red and the two other appear grey in color.



Click on “Return to 3D View” to go back to the normal graphics view.



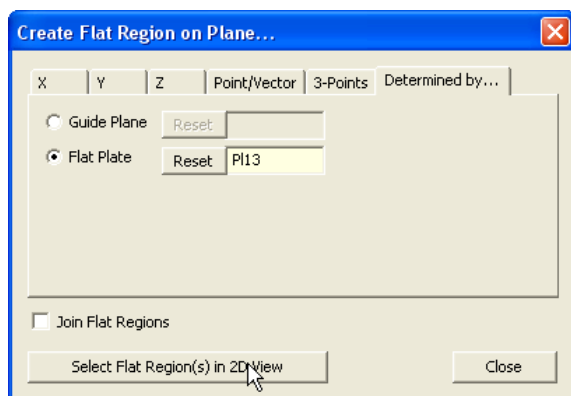
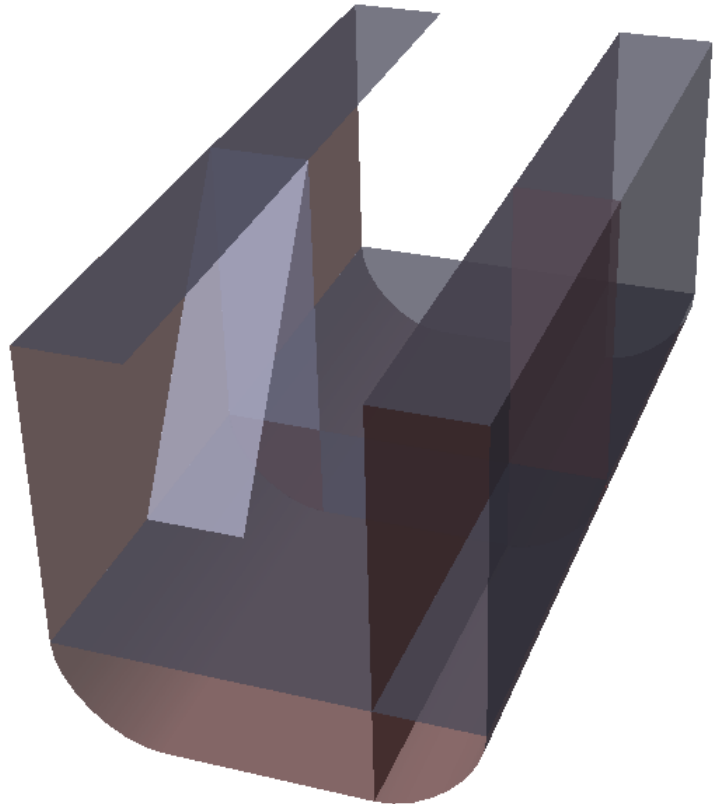
All the five available bounded regions have been clicked, and five plates have been created.



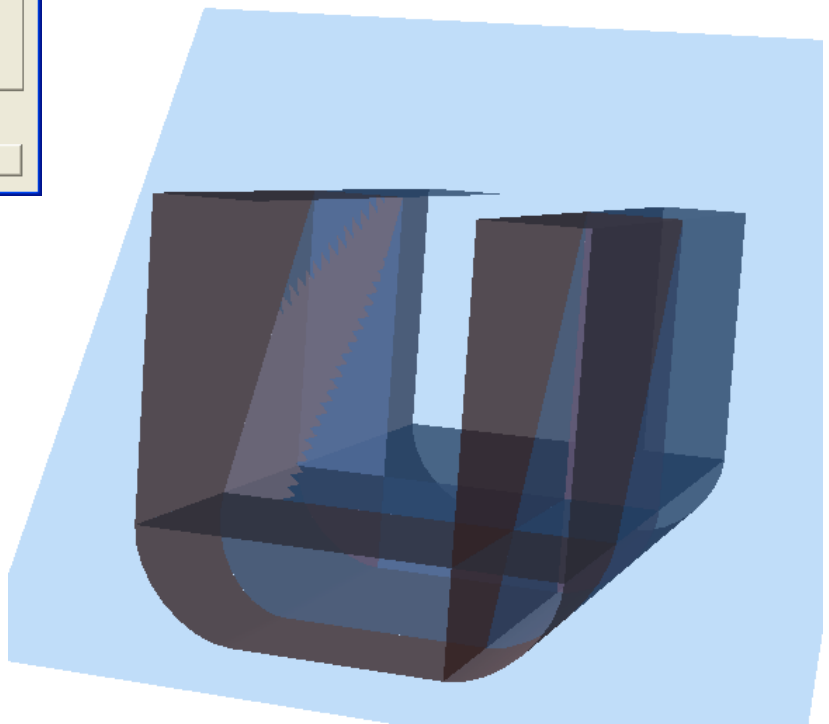
We insert a new plate to demonstrate the possibility for creating plates that are not in the X, Y or Z-plane.

The sloping plate to the left has been inserted. The plate making up the inner side that would have obscured it has been set to invisible.

We open the “Create Flat Region on Plane” again. Unlike in the previous example, we now select the tab “Determined by” at the top of the dialog. We select the radiobutton “FlatPlate” and click on the sloping plate from the previous illustration. The name of the plate appears in the dialog. This time we also choose to check off the checkbox “Join Flat Regions”.



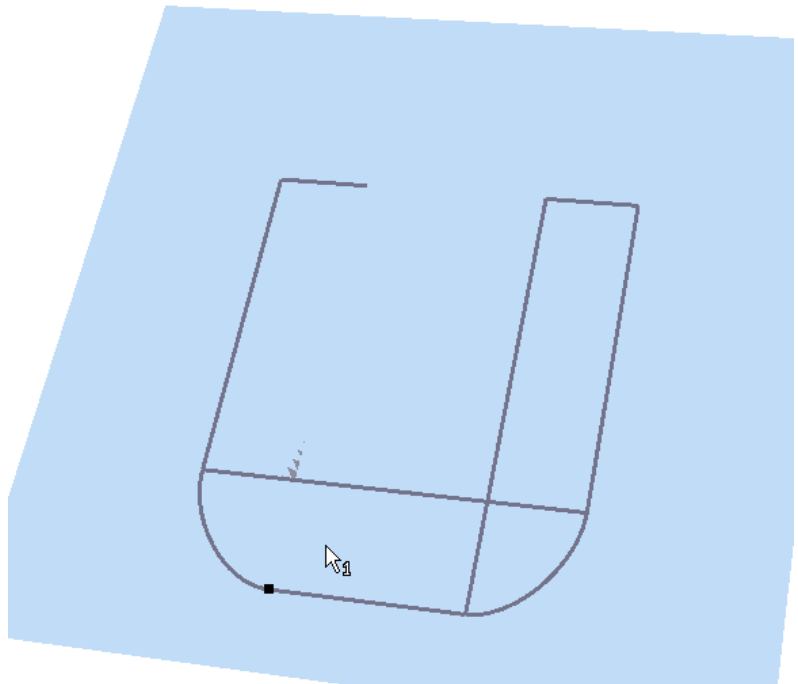
A plane appears in the graphics window.



As can be seen in the illustration, there are a total of three bounded areas that can be used for creating plates.

Note that because the inner side has been set to invisible, one line is missing in this illustration if you compare it with the illustration earlier in this chapter.

We click in both the regions in the double bottom.



Because we had checked the checkbox “Join Flat Regions”, this results in one continuous plate being made.

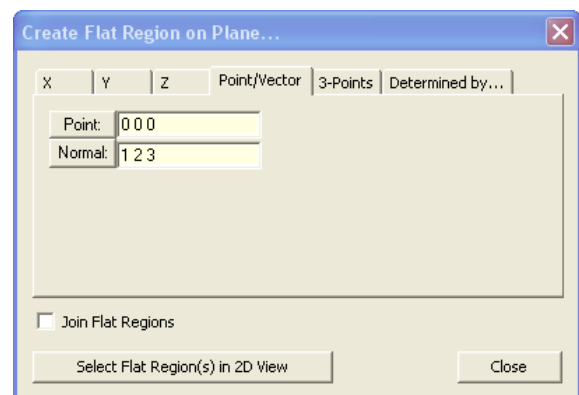
The join operation is always done with the previously created plate. You can reset the stack by checking on/off the "Join Flat Regions" check-box.



In some cases you would like to use an arbitrary plane. This is possible by clicking the “Point/Vector” tab and typing in the coordinates for a point and a normal vector.

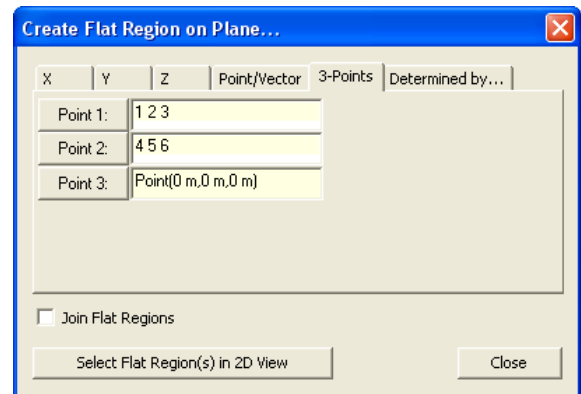
By clicking on the “Point” button you can select a point in the graphics.

By clicking on the “Normal” button you can select a normal vector by clicking two points in the graphics.



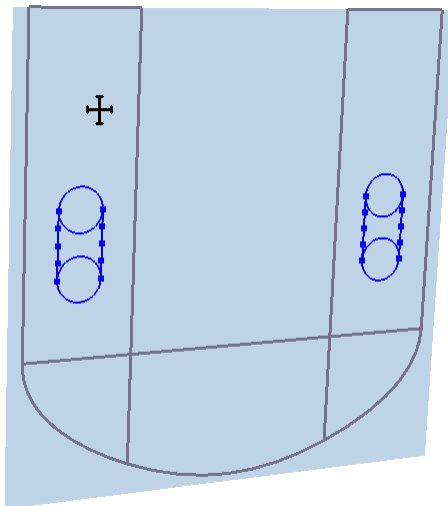
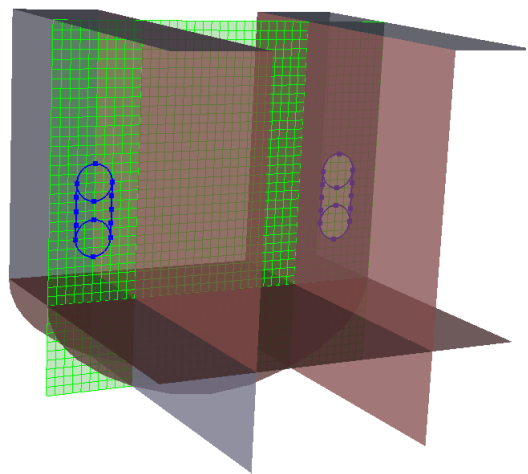
It is also possible to use an arbitrary plane by clicking on the “3 Points” tab.

By typing in the coordinates of the points or selecting them in the graphical view a plane containing the three points is created.

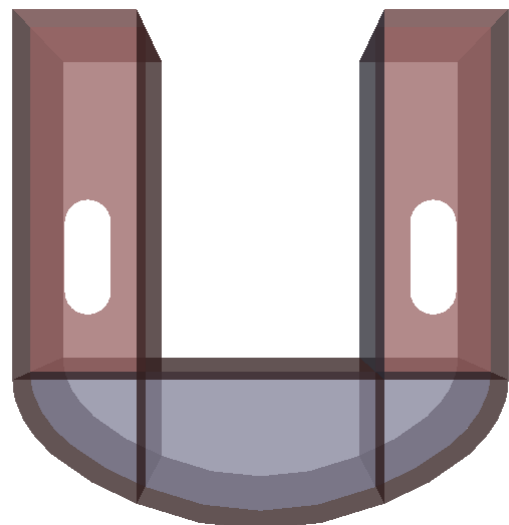


Insert using Flat Region – Manhole example

Creating a plate in a flat region can be used for several purposes. As an example we will here show how you can easily create a plate with manholes. We start with a similar model as in the previous paragraph. We have inserted a guideplane. On the guideplane we have created guidecircles and guidelines to outline our planned manholes.



By clicking once in the area surrounding the manhole, a plate is created. The area bounded by the guide circles and guidelines are left empty, and we have already created a plate with a manhole.

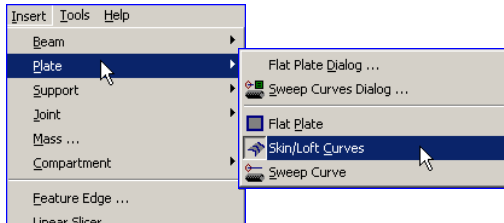


After having clicked all five regions, we get a continuous transverse bulkhead, complete with two manholes.

3.3.1.4 Insert using loft

Lofting is a variant of skinning involving both shells and curves to control the shape of the new surface to be created. The operation takes advantage of coinciding shells so they become G1 continuous (G1 continuity is where tangent lines are smoothly connected). This feature may be used to create surfaces where there is a varying shape (e.g. fore and aft part of a vessel) as compared to surfaces with a constant shape (typically mid-ship parts).

The feature is accessed from **Insert/Plate/Skin/Loft Curves** or from the tool button:



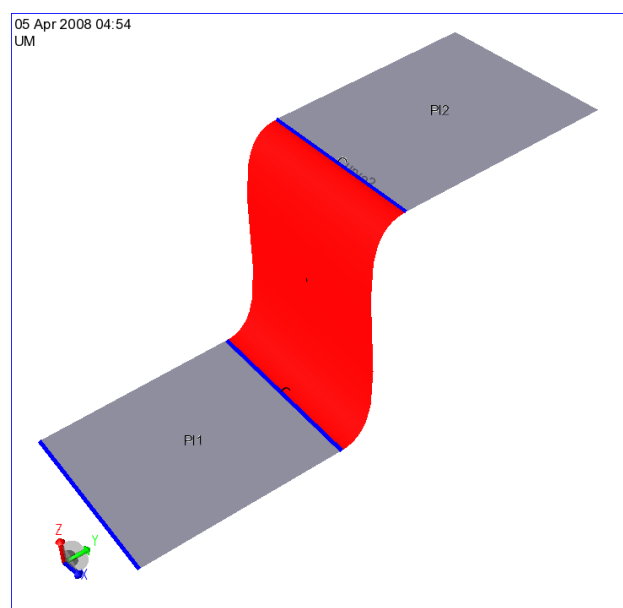
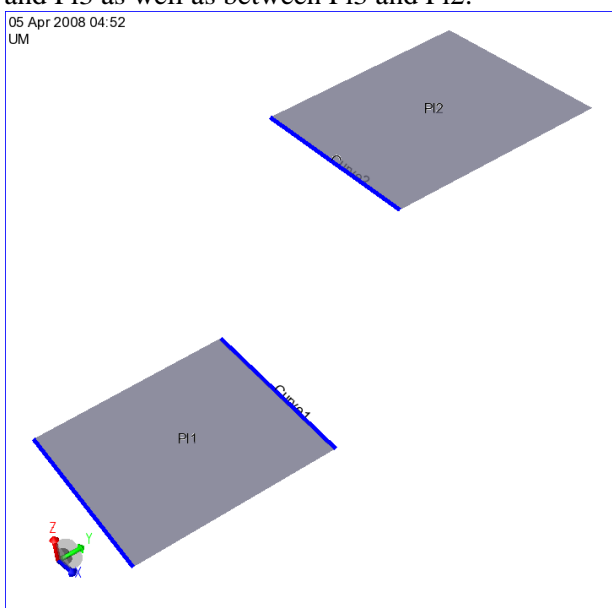
In a pure skin curve operation the new surface is created from referring curves only. In lofting the procedure is somewhat different as the selection normally contains

- a surface as a start condition
- several curves, the first curve must coincide with the start surface and the last curve must coincide with the end surface
- a surface as a stop condition

It is also possible to start or stop with a curve, but lofting must include at least one surface. In the following some examples will be given on how to use lofting.

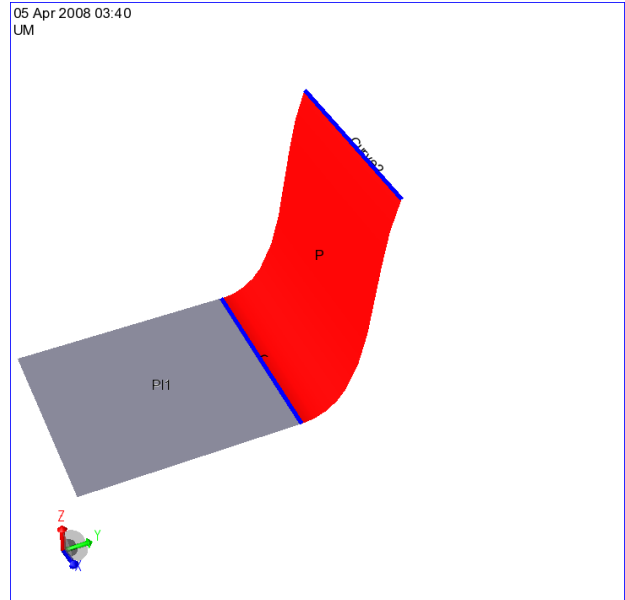
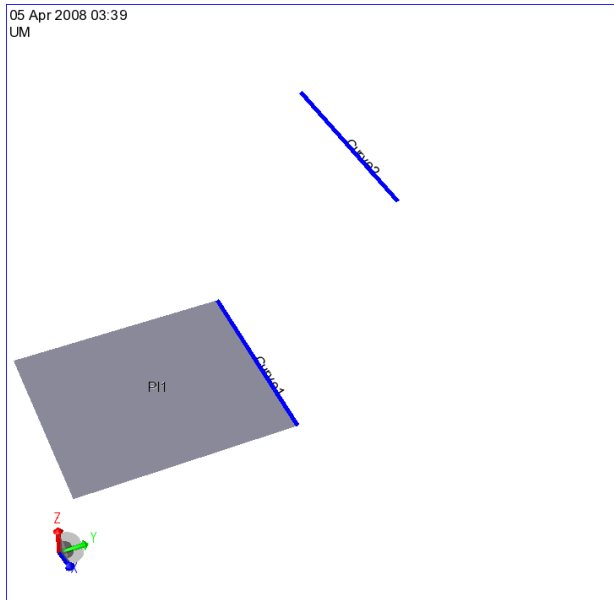
Make a shell from a plate, two curves and a plate.

In this case the new surface, P13, has been created from selecting in sequence P11, Curve1, Curve2 and P12. Single-click the last object of the lofting operation when this object is a plate, in this case P12. As in the case of pure skinning, the selected items are highlighted in orange colour. There is G1 continuity between P11 and P13 as well as between P13 and P12.

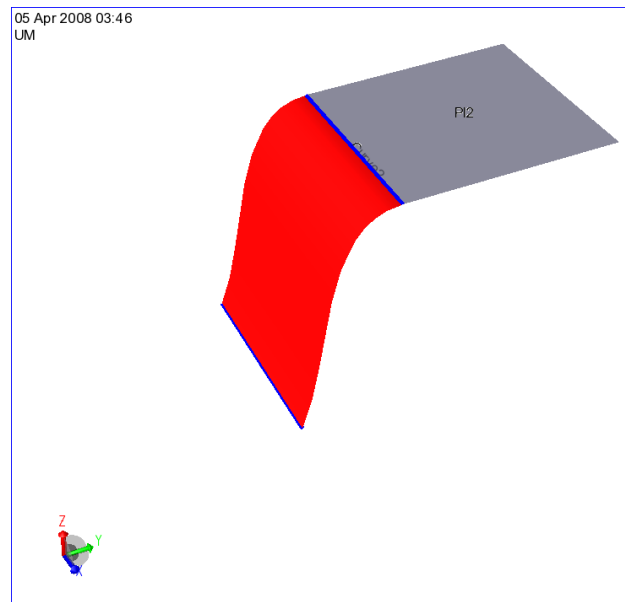
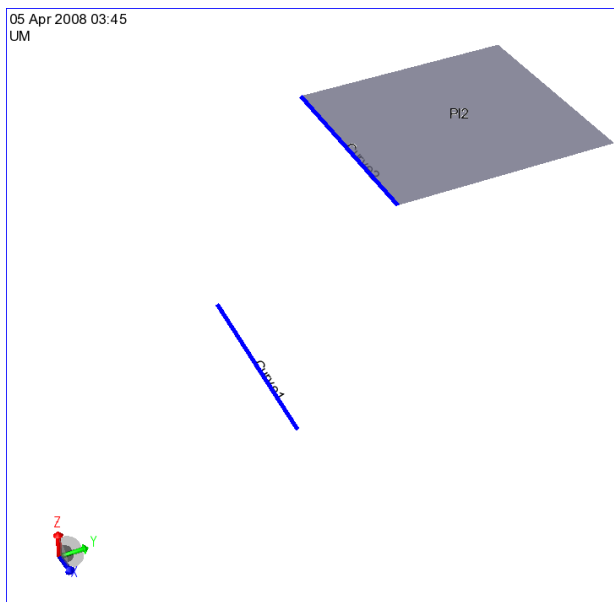


Make a shell from one plate and two curves.

The shell P13 is created using lofting by selecting P11, Curve1 and Curve2 in sequence. Double-click the last object of the lofting operation when this object is a curve, in this case Curve2. There is G1 continuity between P11 and P13.

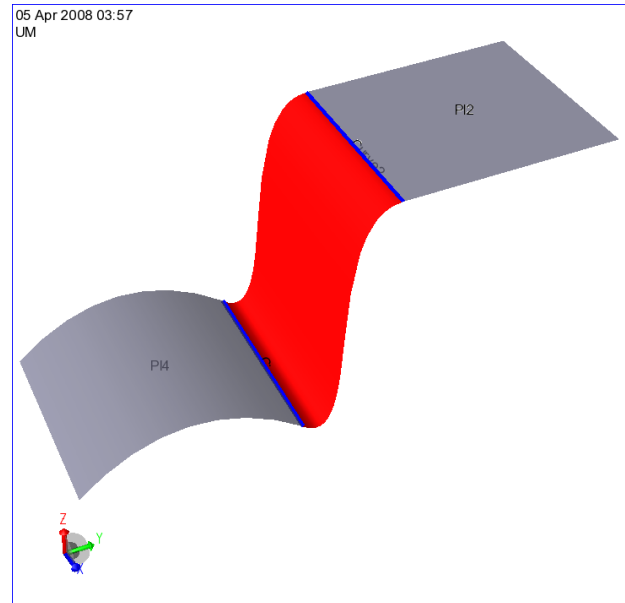
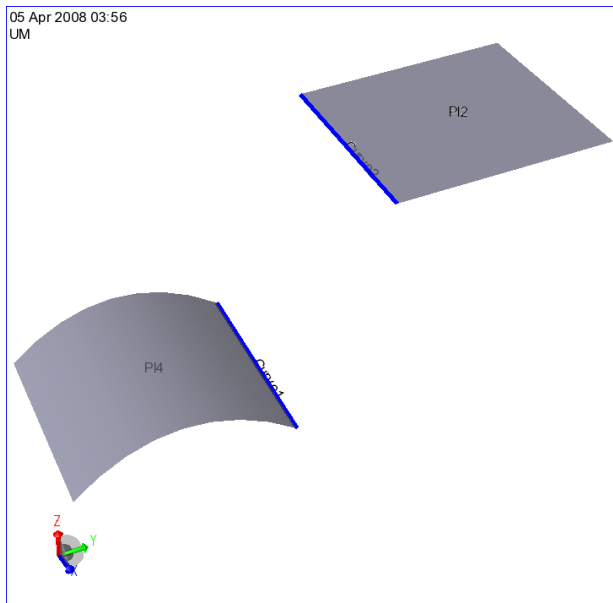
***Make a shell from two curves and one plate.***

This example is basically the same as above, but in this case the selection is in opposite sequence: Curve1, Curve2 and P12. Single-click P12 to end the lofting. There is G1 continuity between P13 and P12.

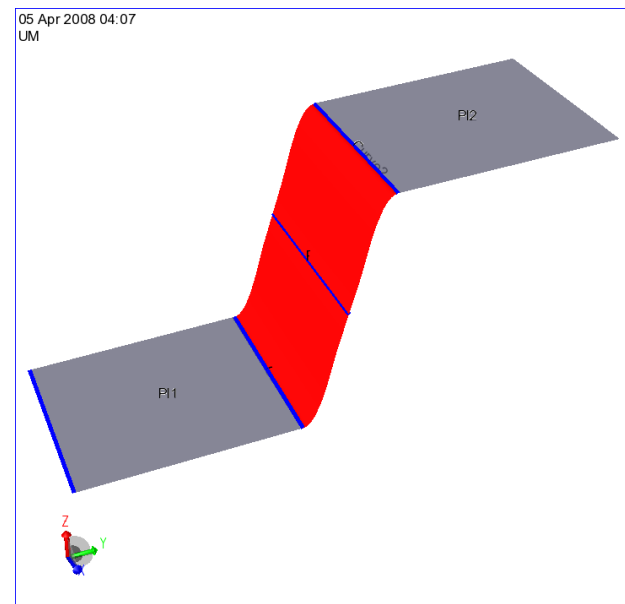
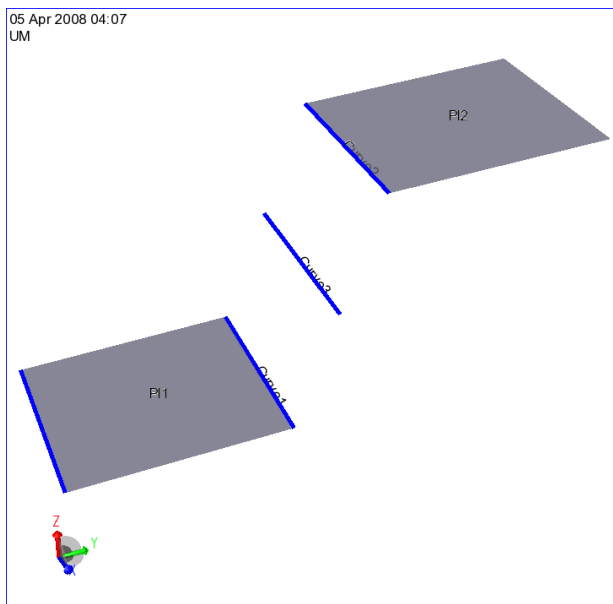


Make a shell from one shell, two curves and a plate.

It is also possible to use shells (i.e. curved plates) as start and stop conditions in lofting. This example shows the G1 continuous shell P15 created from (in sequence) the shell P14, Curve1, Curve2 and plate P12.

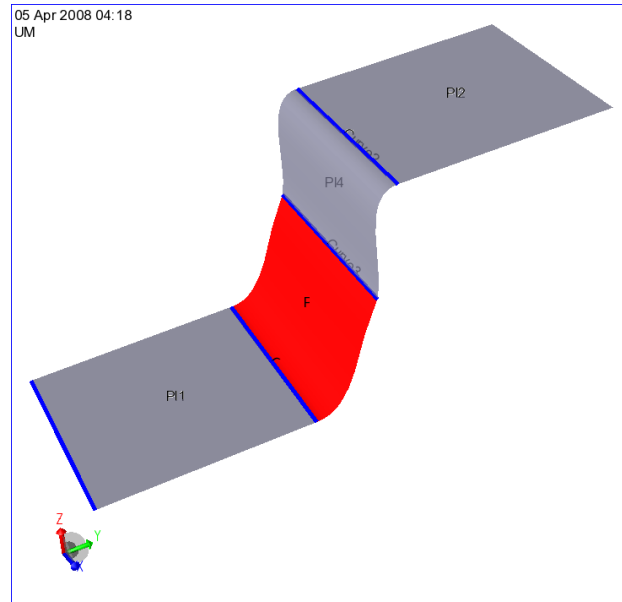
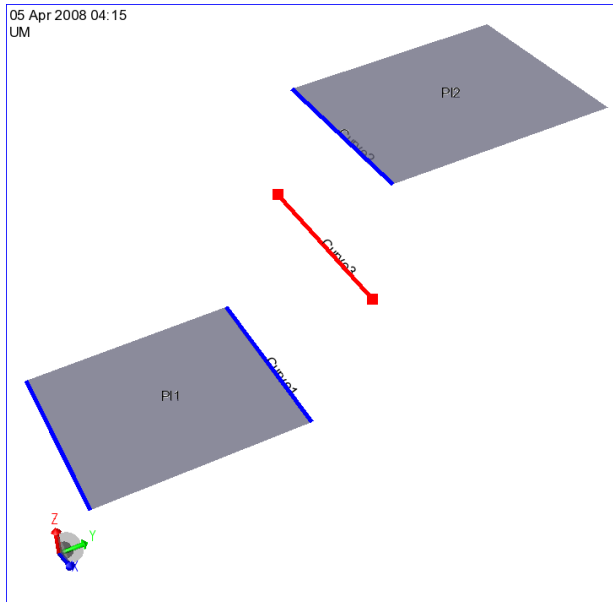
***Make a shell from one plate, three curves and a plate.***

In this example the new shell P13 is G1 continuous with P11, but not with P12. The lofting sequence is P11, Curve1, Curve3, Curve2 and P12. Curve3 is halfway between Curve1 and Curve2 in this case.

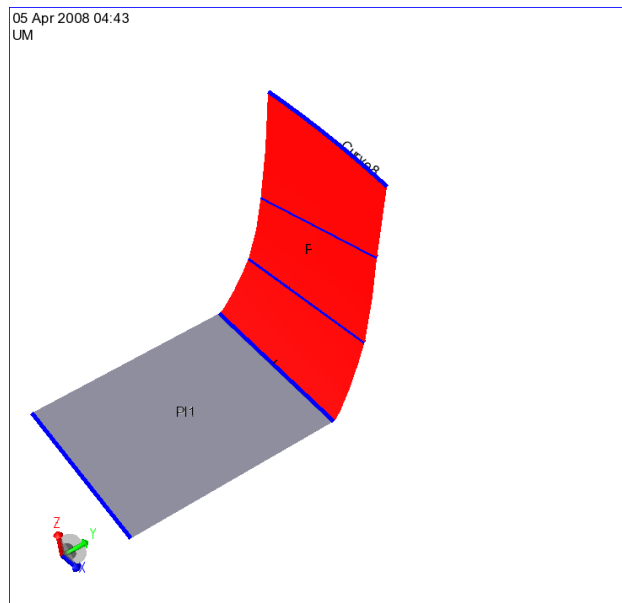
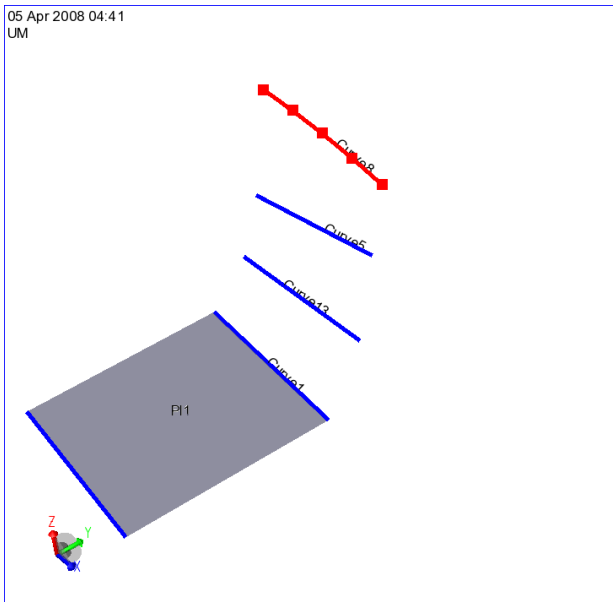


Make two shells from one plate, three curves and a plate.

The two new shells P13 and P14 are made G1 continuous with their neighbour plates P11 and P12, respectively, by using a curve as stop and start condition. The shell P13 is created from lofting P11, Curve1 and Curve3, while the shell P14 is built from Curve3, Curve2 and P12. Note that there is no G1 continuity between P13 and P14. In this case Curve3 is not halfway between Curve1 and Curve2. P13 is highlighted below.



Make a shell from one plate and four curves.

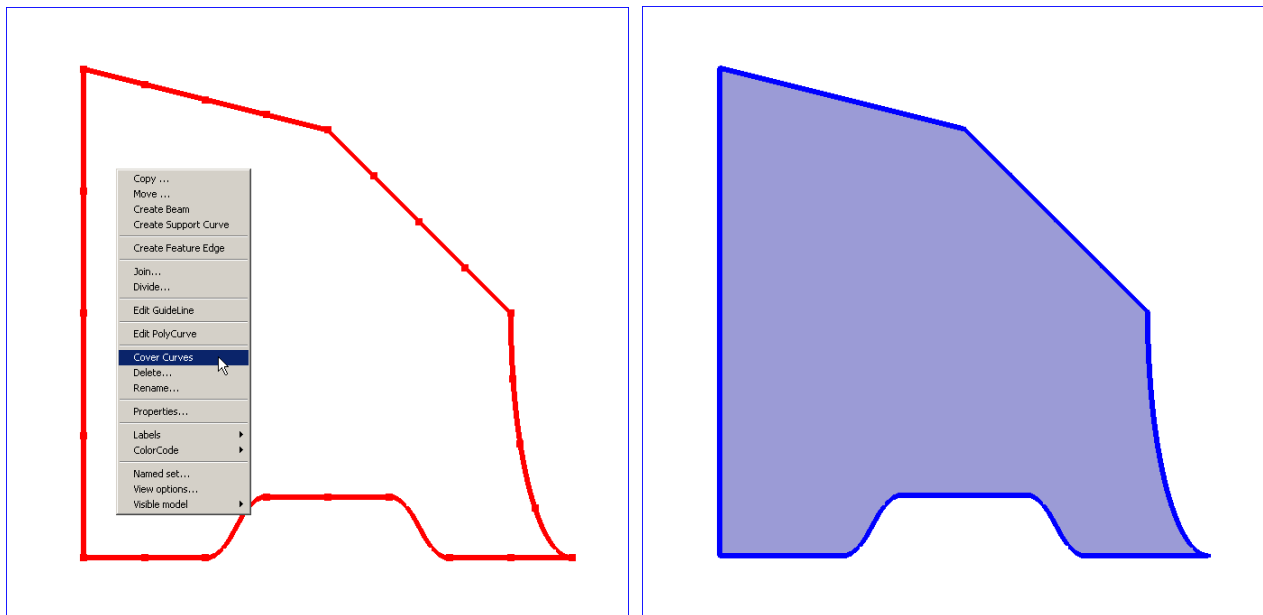


In this case the middle curves Curve13 and Curve5 are not parallel with Curve1. Furthermore, Curve8 has a curvature (highlighted above). As such, this is an example relevant for modelling of hull forms in the fore and aft part of a floater. The new shell is built using the sequence P11, Curve1, Curve13, Curve5 and Curve8.

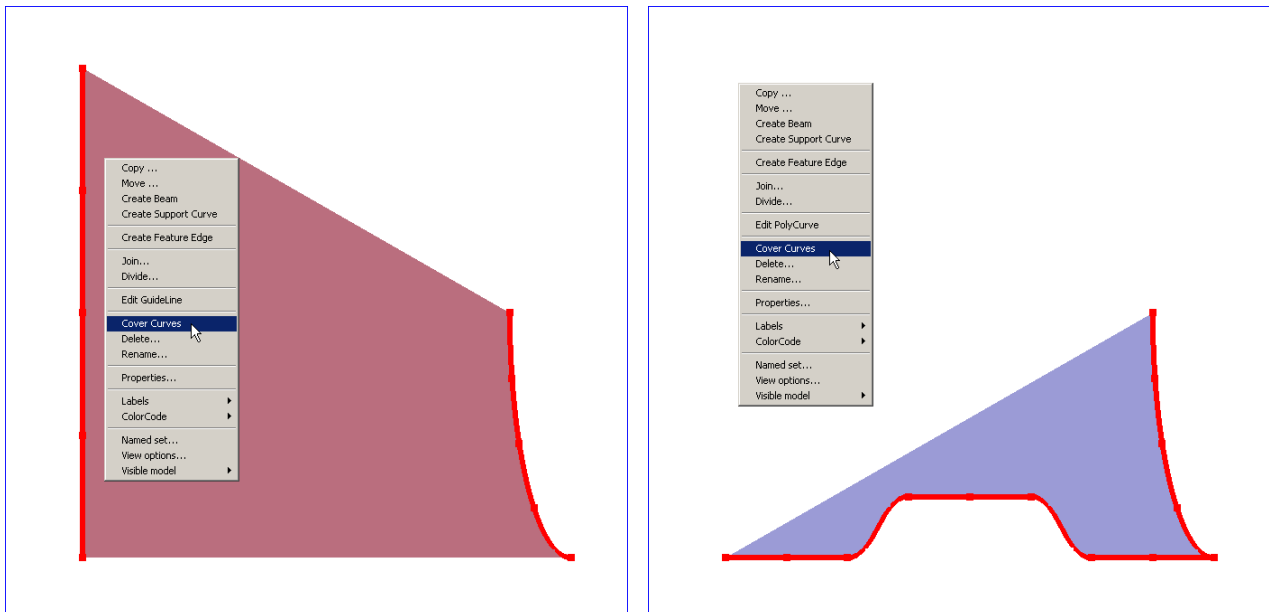
3.3.1.5 Insert using cover curves

The term cover in this context mean fill between curves. The curves may form closed loop of curves or they may consist of two individual curves – in the latter case, temporarily straight lines will be set up between the end points. There are five examples to show how cover curves can be used

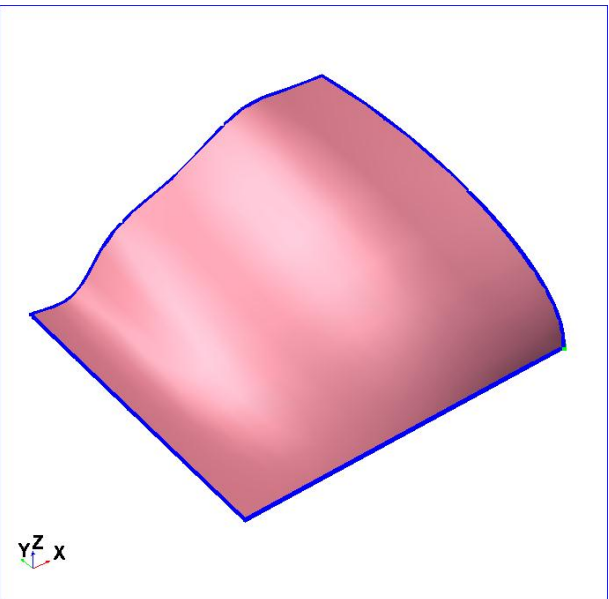
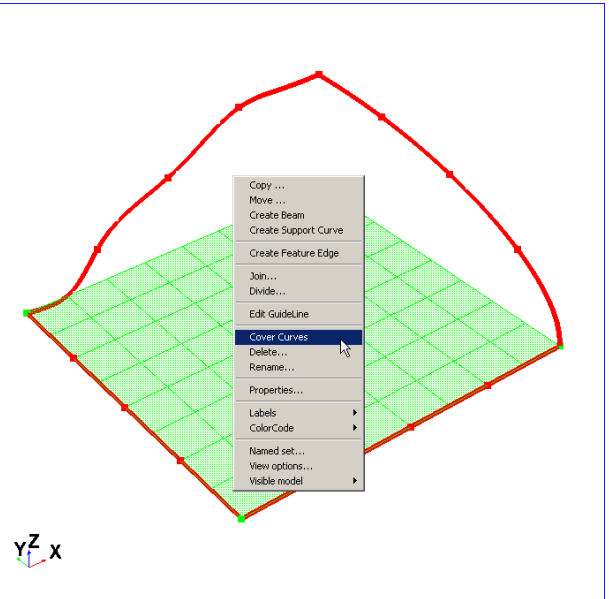
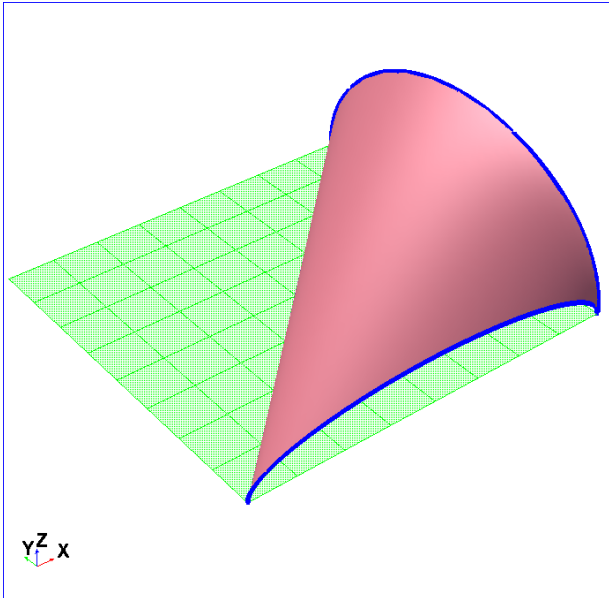
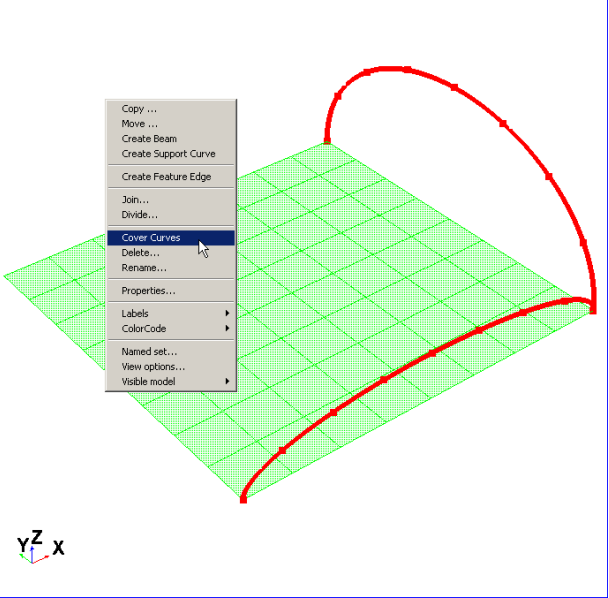
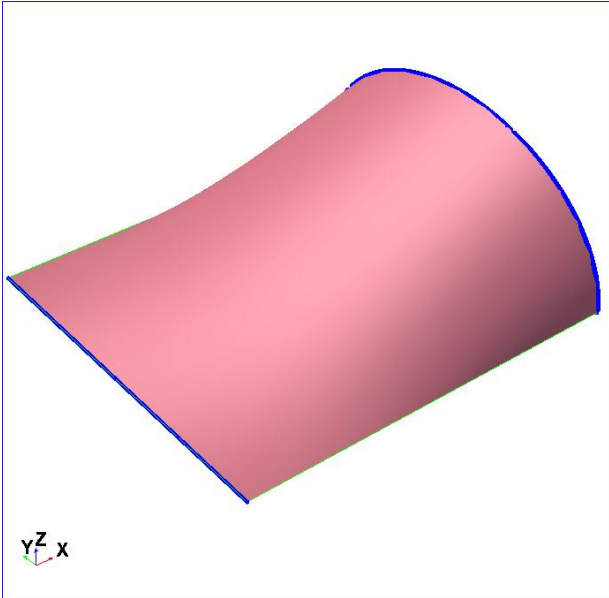
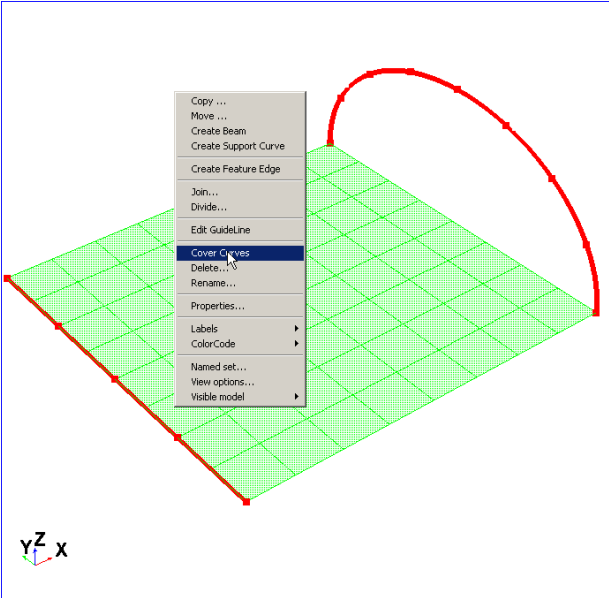
The cover curve command is available from the context sensitive menu (select objects, **RMB** and *Cover Curves*). In this example, there are several curves that form a closed loop. The cover operation will fill the surface defined by the closed loop.



The examples below show the cover curve operation on two selected curves. As can be seen, straight lines are used to make a closed loop. Notice that the each must be continuous to perform the below operations

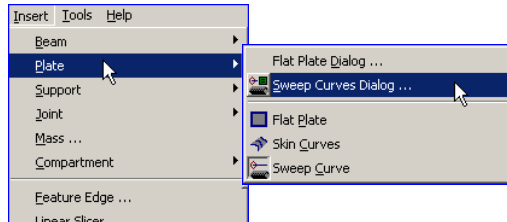


The above examples assume that the curves are created in a flat plane. For curves defined in different planes, GeniE will ensure that the surface to be filled lies in the continuous plane, typically a part of a sphere. Some examples are given below how to do this. Observe that GeniE can not handle all type of constellations. If you experience problems, you should divide your structure into smaller parts.



3.3.1.6 Insert using extrude

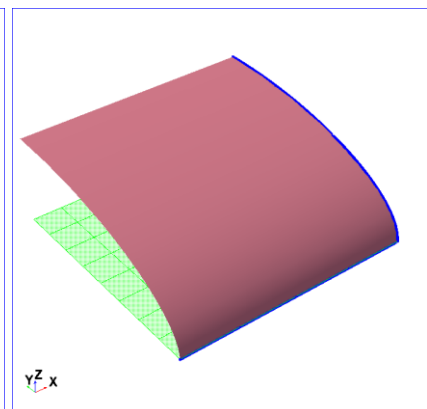
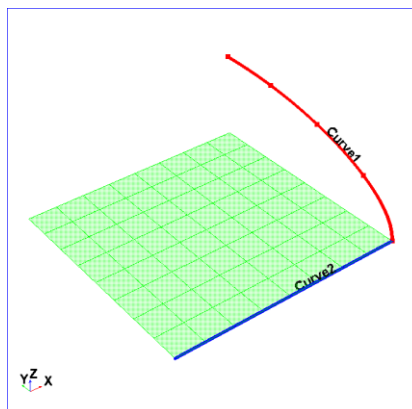
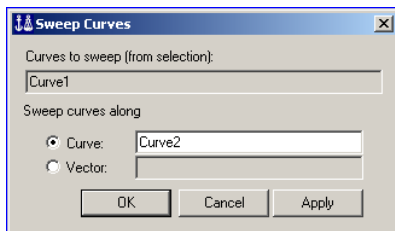
This command is available from the **Insert/Plate/Sweep Curves Dialog** or from tool button.



The sweeping is done using one curve to sweep along another curve. They can both be curved. Some examples are shown below, and notice GeniE is not capable of making a surface for all constellations of curves. If you experience problems, you should split up your structure into minor parts.

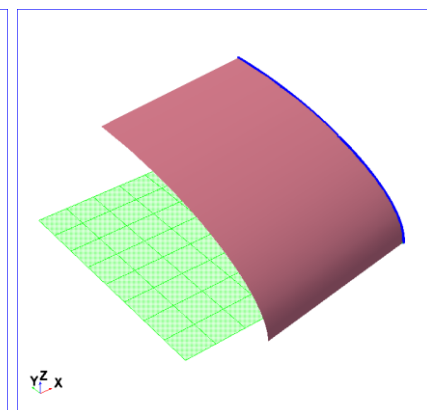
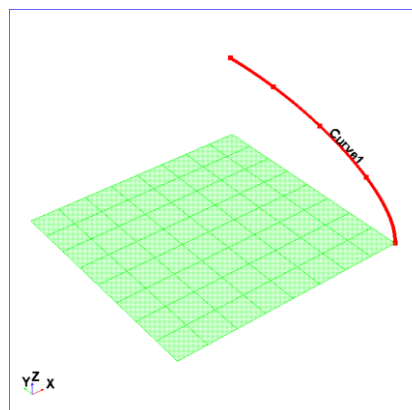
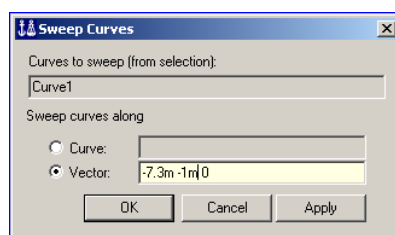
The extrude functionality can be used among others when you want to make structure based on 2D parts. Examples of such may be the longitudinal and parallel parts of a semi-submersible pontoon or a ship hull based on a web-frame. How to create 2D parts and extrude these into 3D structures is explained later in this user manual.

The first example shows how to do it using the **Insert/Plate/Sweep Curves Dialog**. Select the curve you want to sweep and open up the sweep curve dialog

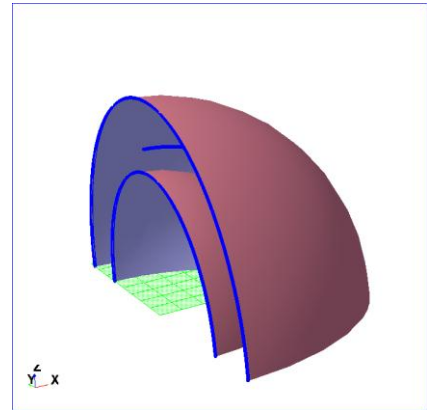
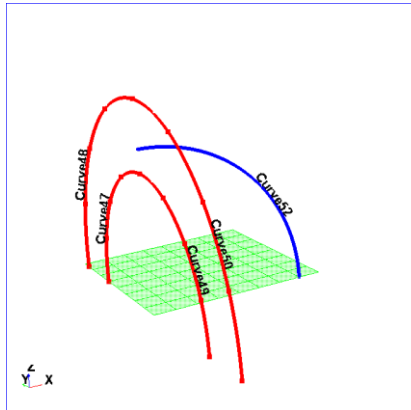
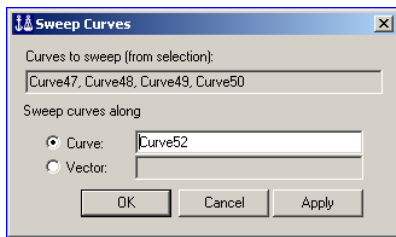


Curve1 is automatically filled in the dialog above (you may select more than one curve). The curve to sweep along must be filled by clicking on the curve (or typing in the name); in this case *Curve2*.

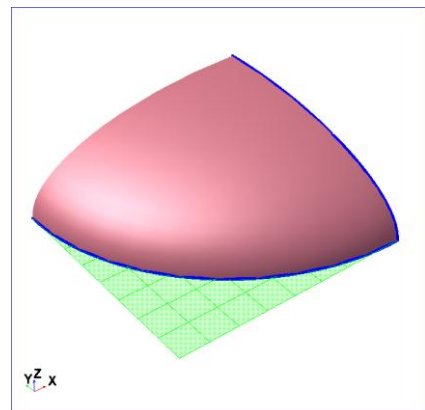
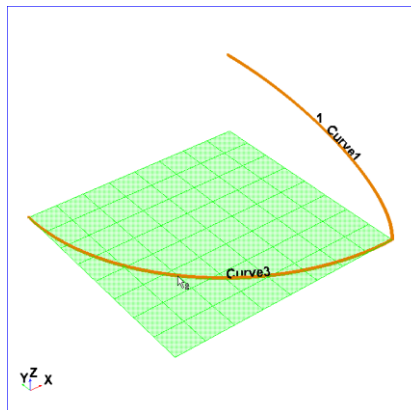
The pictures to the right shows sweep using a vector. The vector can be manually typed in or from the graphic window.



In this example more than one curve has been selected for sweeping. The four curves *Curve47* through *Curve50* have been selected. These are automatically filled into the sweep curve dialog. *Curve52* is specified as the sweep curve.

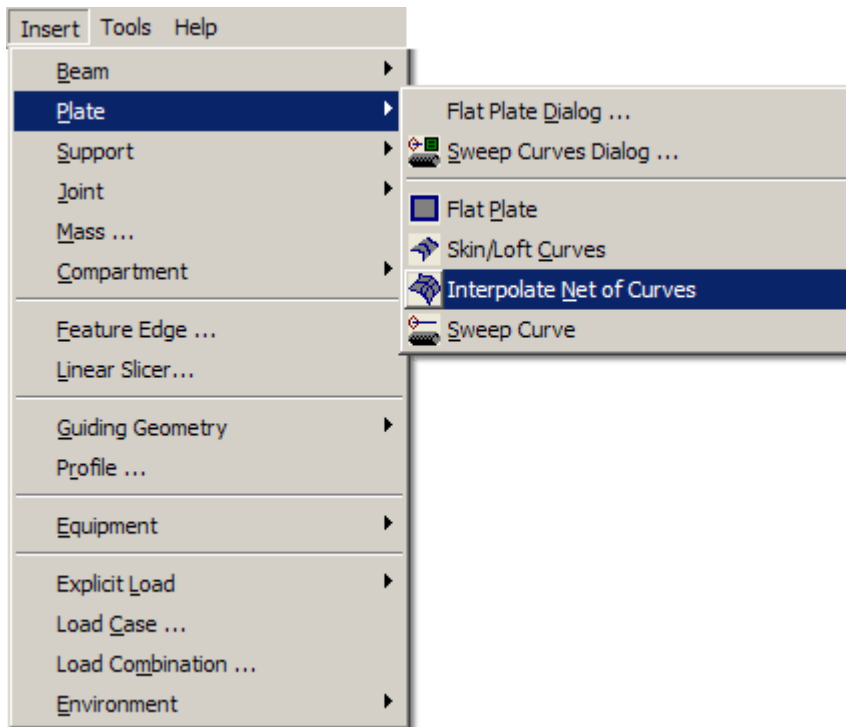


When you use the tool button the first input is the curve to be swept, while the second input yields the curves to sweep along. In the example to the right Curve1 is clicked first (and denoted "1" in the graphics) while Curve3 is clicked thereafter.



3.3.1.7 Insert using curve-net interpolation

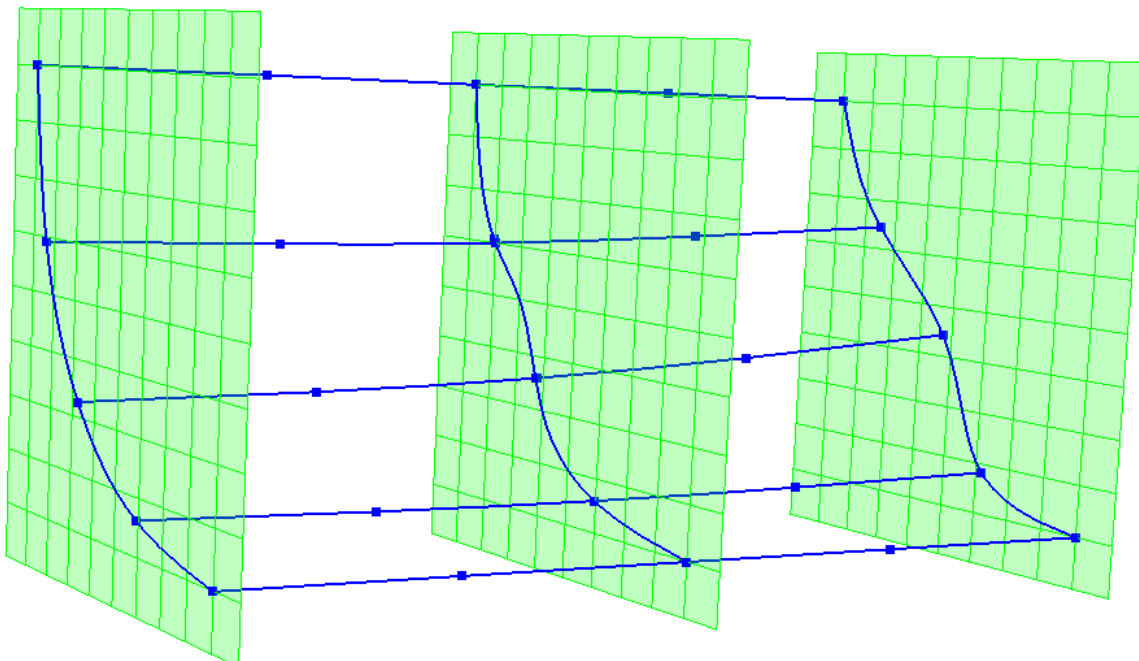
Curve-net interpolation constructs a surface that interpolates two groups of curves. The curves in the first group must cross the curves in the second group – thereby forming a net. The curves need to be connected (intersect) at the corners, but no common end point is required.



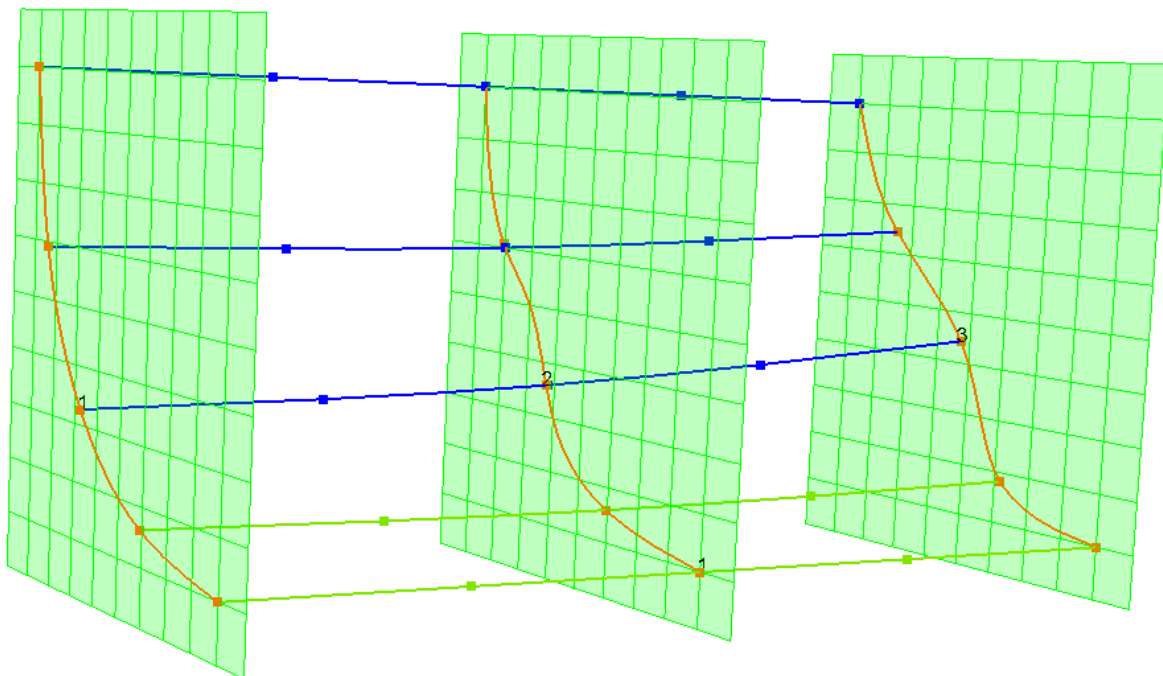
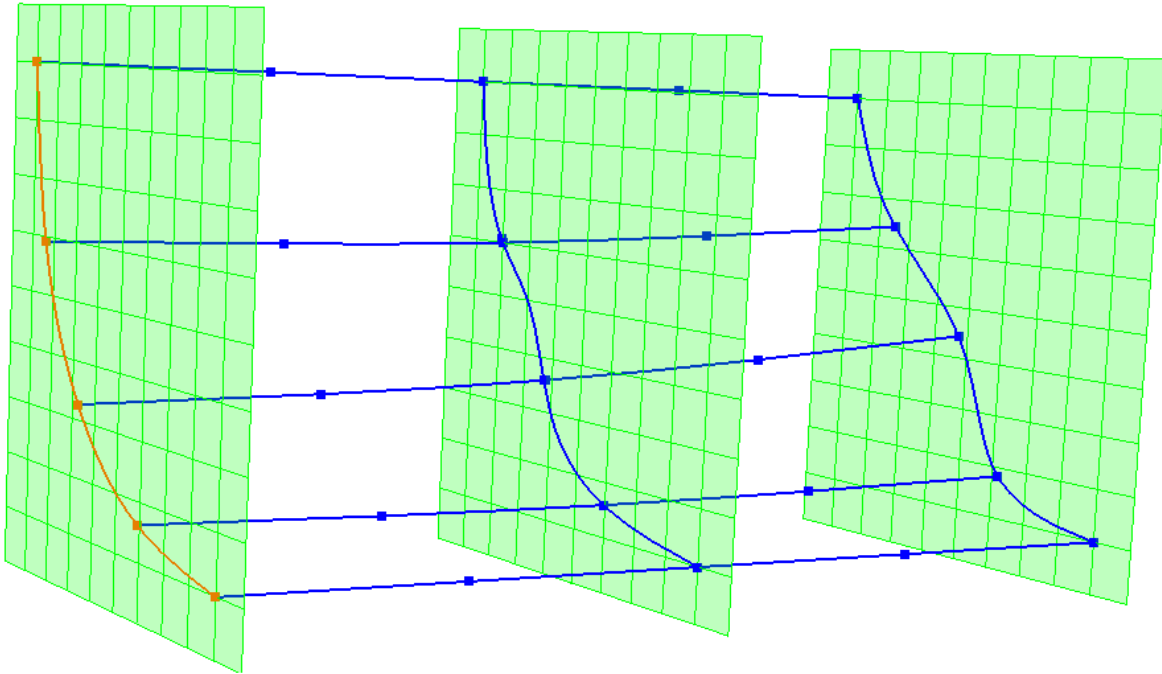
When using the curve-net interpolation you need to select (click on) which curves form the two groups. You must first select all curves in one direction (group one). Stop the input for this direction by double click the last curve. Then select curves in the other direction (group two). Double click the last curve to complete the command.

Please note that when selecting the curves for the group they must be given successively (i.e. in the order they are located).

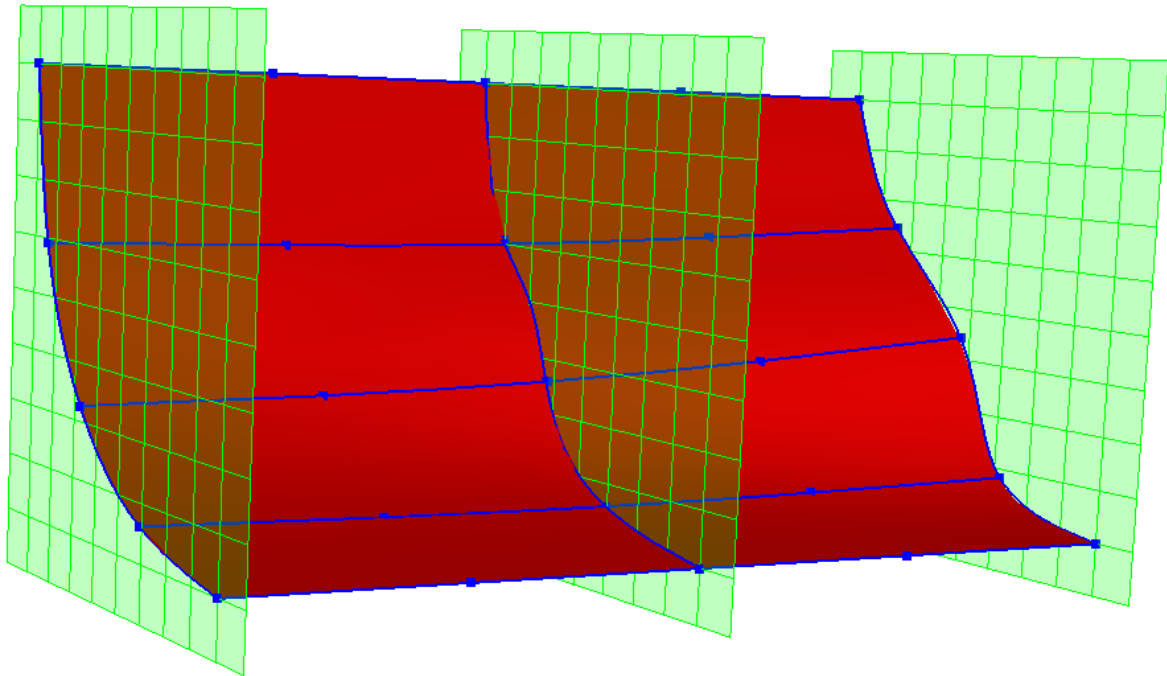
Typically, the curve-net of three ($N=3$) and five ($M=5$) curves will look like the following figure.



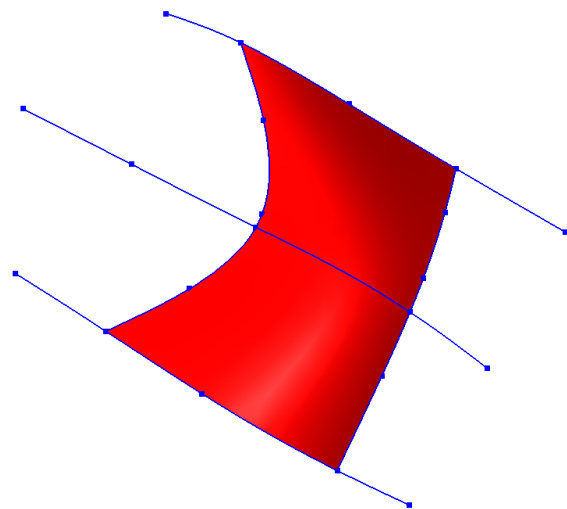
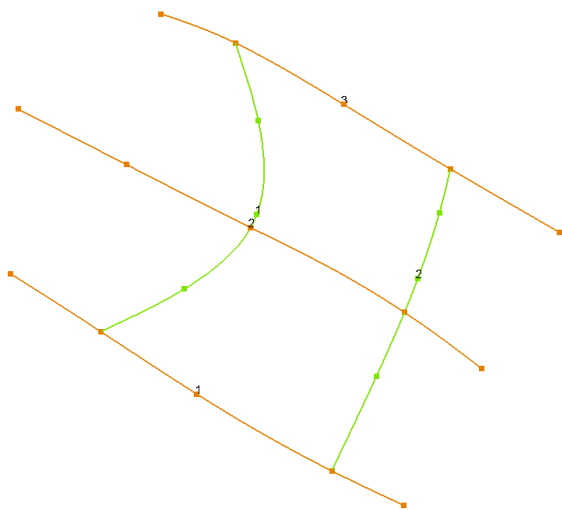
When moving the mouse over the curve it becomes orange for the curves of the first group and green for the curves of the second group. A selected curve stays orange/green and is denoted by the sequence number.



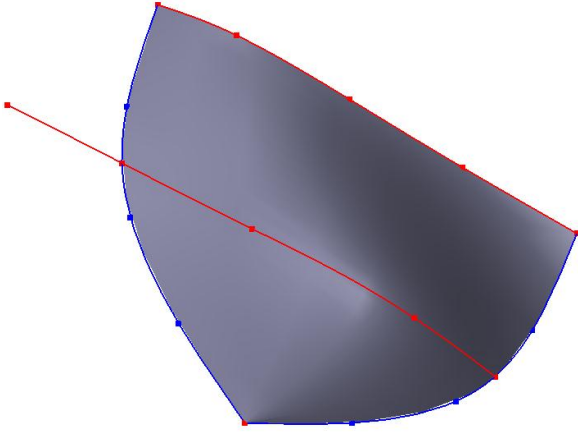
The above picture shows that three curves have been added in one direction and one in the other direction, a second is due to be selected. The illustrations below show the plate, as a result of the curve-net interpolation, when all three and five curves have been selected, from two different points of view.



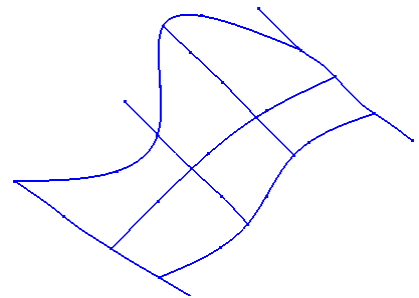
Remarks: The method constructs surfaces that cover only the rectangular patches, formed by the curves without interpolating them necessarily (see next figures).



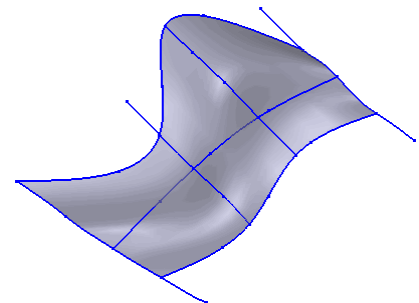
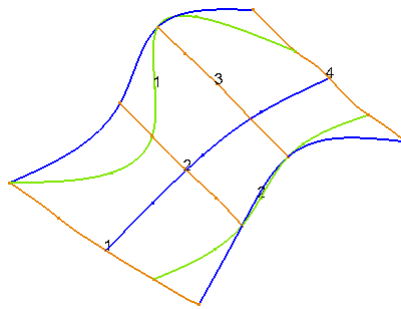
The method can be applied to curves that degenerate to points thus forming triangular patches. Note that the point should be explicitly given as “guide-point”. In the next figure the lower curve in one direction is a single (red) point, which forms the triangular patch along with next red curve and the (blue) curves in the other direction, which intersect at this point.



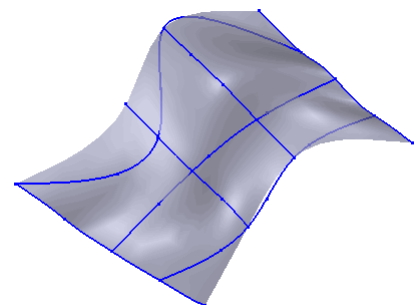
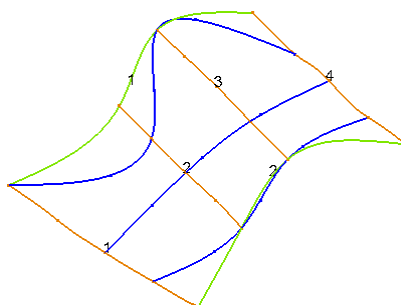
After you have selected the first group of curves, the program may suggest additional curves that you can use to make your second group of curves. The example net to the right is used to illustrate this:



The system suggests two additional curves. The user chooses to use the original curves. The resulting curved shell is showed to the right.

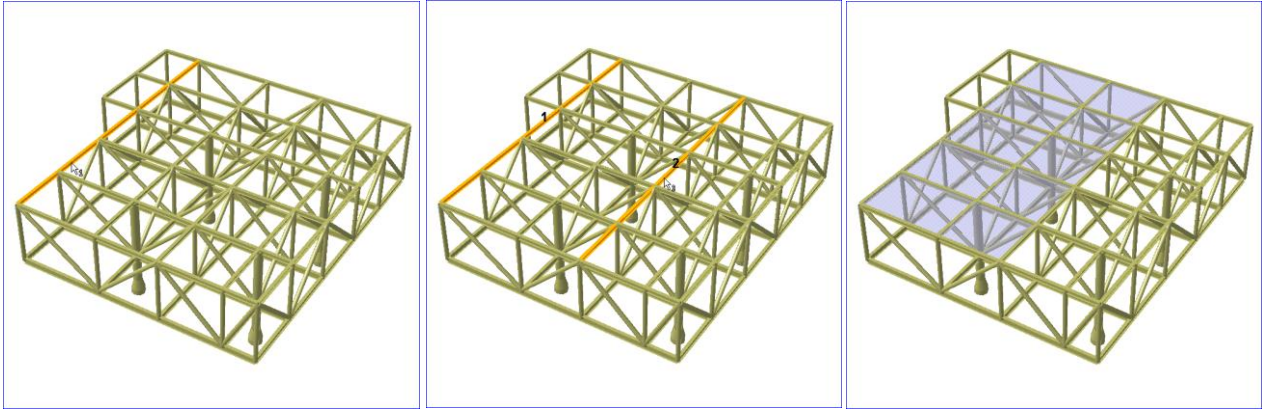


The system suggests two additional curves. The user chooses to use the suggested curves. The resulting curved shell is showed to the right. Note the differences between the two resulting curved shells.



3.3.1.8 *Define plates using beams as reference*

It is also possible to create plates and shells by referring to beams in stead of guide curves. It works the same way as for curves. One example is shown below illustrating the use of beams in a skin operation.

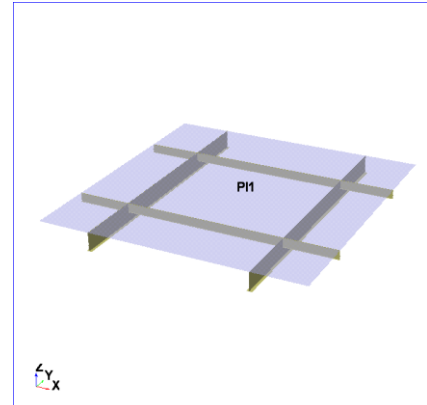
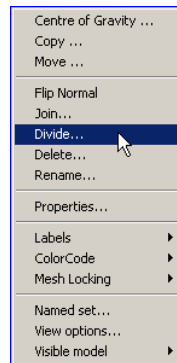


3.3.1.9 Divide using existing structure

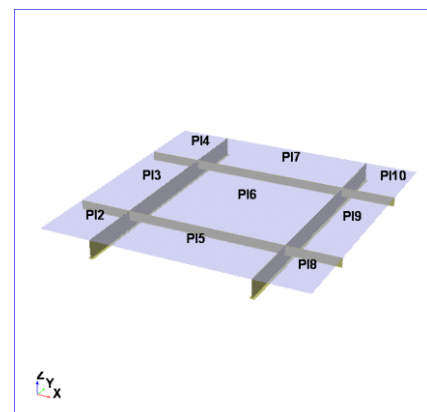
When using the top-down modelling approach it will often be necessary to divide the structure. The following example shows how to divide a plate (it is similar for beams and stiffeners) based on intersecting structure (plates or stiffeners).

The divide function is available from select object(s), **RMB** and *Divide*.

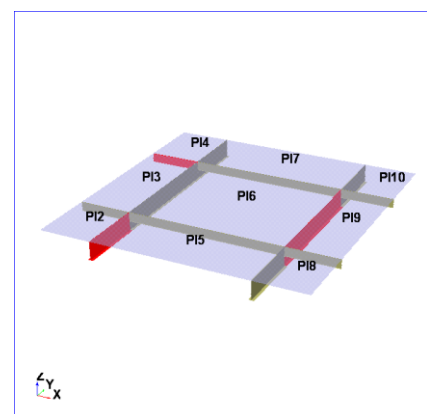
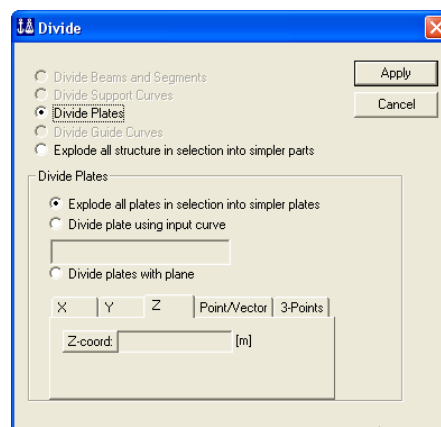
This example has a plate (*PI1*) intersected by 4 stiffeners. Select the plate, **RMB** and choose *Divide*.



By using the option *Divide Plates* (first radio button) and using *Explode all plates in selection into simpler plates* (second radio button), the plate PI1 is divided into 9 new plates.



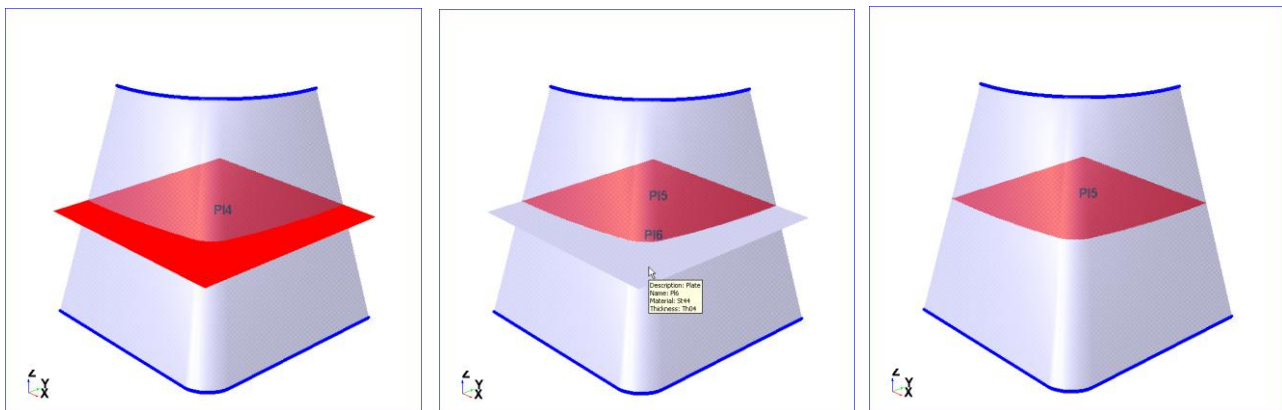
In case the stiffeners were part of the selection both stiffeners and plates are split into minor parts depending on the intersection pattern.



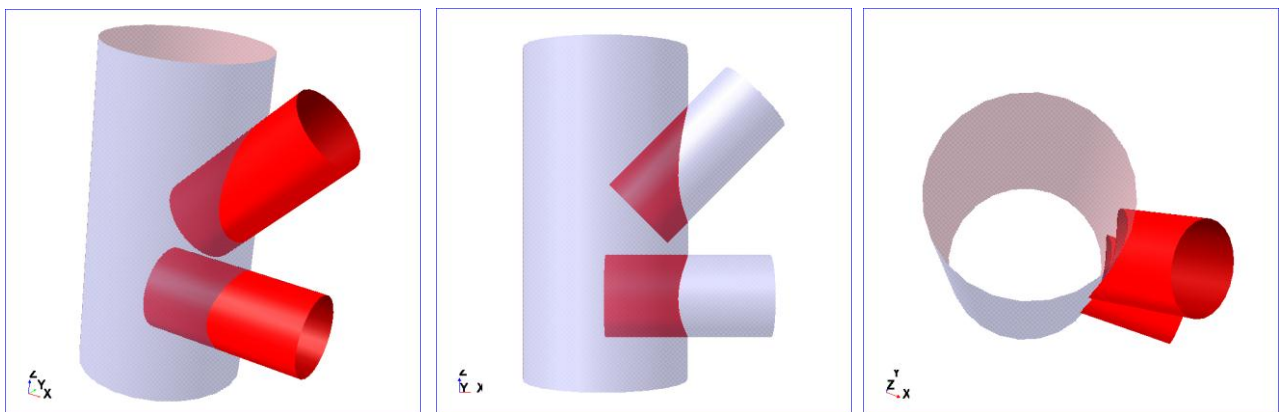
3.3.1.10 Divide and trim using existing structure

To trim a structural part to another it is necessary to divide first and then delete the superfluous parts. Two examples of such are shown.

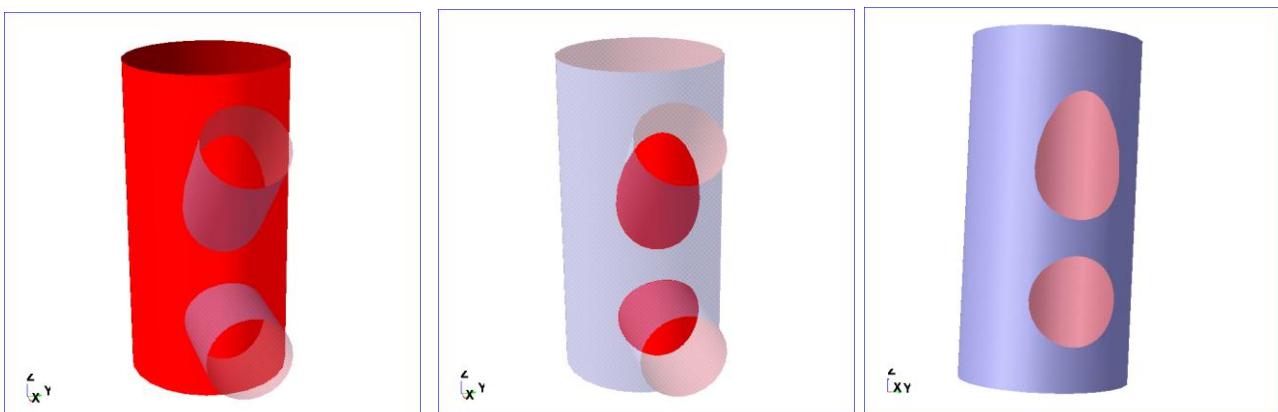
In the first case a horizontal plate (*P14*) is intersecting a conical transition part (like for example at the transition between a semi-submersible pontoon and column or in a crane pedestal). Select the plate *P14*, **RMB** and *Divide* using option explode all plates. Plate *P14* is now split in two parts, *P15* and *P16*. Select plate *P16* and delete. You have now a horizontal plate trimmed to the conical transition part.



Another example is when tubes are intersecting another tube, typically in tubular joint. This example shows two tubes intersecting another tube. Select both incoming tubes, **RMB** and *Divide* using option explode all plates. Select the inner parts of the tubes and delete them. The tubes are now trimmed to the column.



If you want to remove the plugs inside the incoming tubes, repeat the process by dividing the column and deleting the superfluous parts. In the bottom right picture, the incoming braces have been removed for visibility.



3.3.1.11 Divide using guide curves

This example shows how to use a guide curve to divide a plate. Select plate *PI1*, **RMB** and *Divide*. Choose divide using input curve and specify *Curve1* (from graphics or manually).

Plate *PI1* is now divided into two and the new plate *PI2* is created.

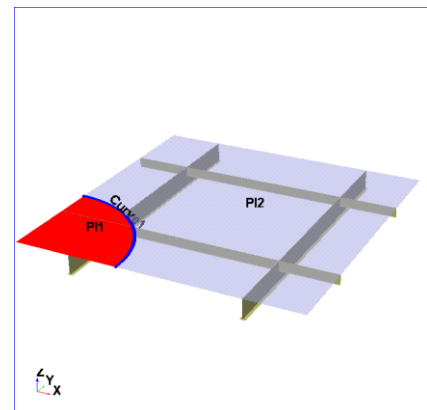
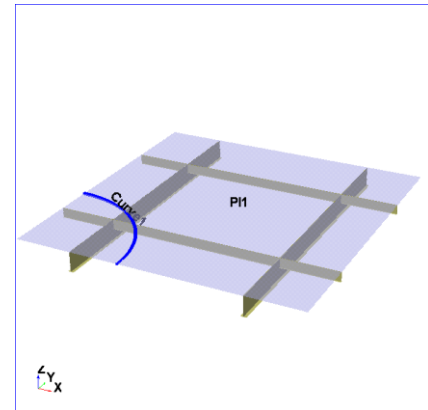
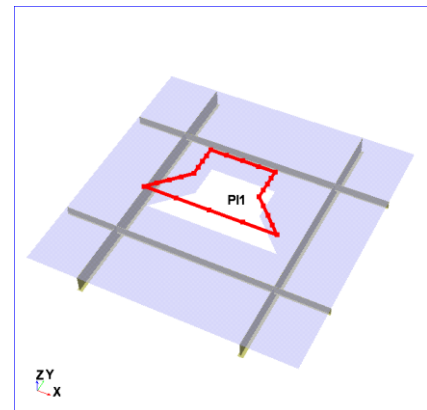
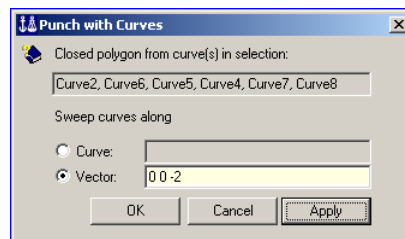
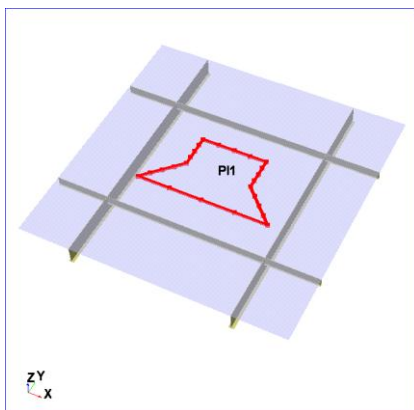


Plate *PI1* may now be deleted if you want to create a cut-out.

3.3.1.12 Punch using guide curves

The above example assumed divide using one continuous line. In the example below a plate is punched (divide and trim) using multiple curves. This operation is available from **Tools/Structure/Curve Punch**. Notice that the curves can not be in the same plane as the structure to be punched; in this case the curves are 1m above the plate.

Select the curves that form a closed loop, open the *curve punch* menu and give a vector. Notice that the solid that will be created as a result of the curves and the vector will remove all structure inside the solid.



This technique is often used when making holes in web-frames using profiles with fillet curves.

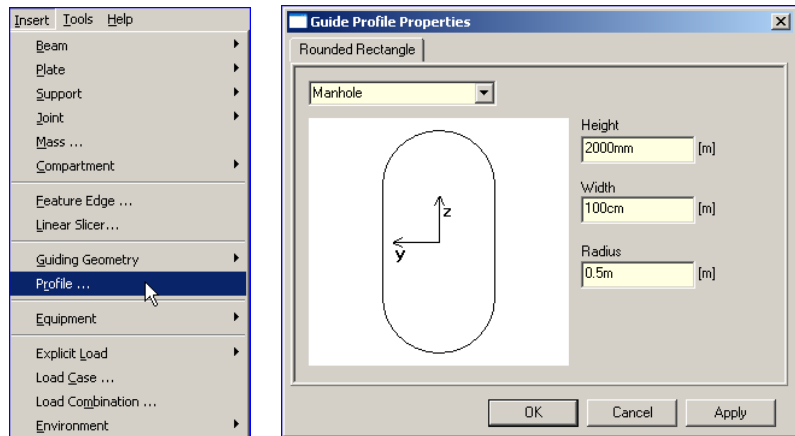
The plate that has been punched keeps its name. Beams, however, are normally divided into several beams.

3.3.1.13 Punch and divide using a profile

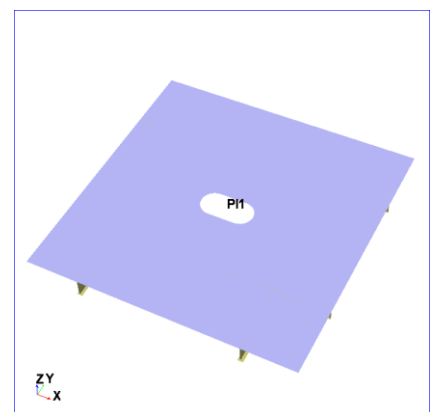
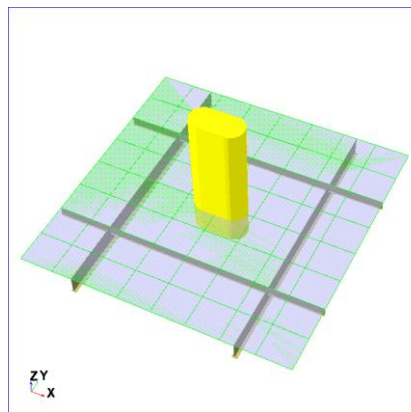
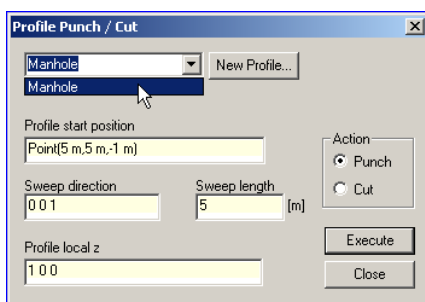
If you have a profile that varies from a square to a circle you can use the option **Tools/Structure/Punch**. Prior to the punch operation it is necessary to define a punching tool. This can be done from inside the punching dialog or from **Insert/Profile**. In the example below a profile with rounded corners (*Manhole*) is used to punch and divide a plate (*Plt*).

The profile *Manhole* is created with length 2m, width 1m and radius of corners 0.5m.

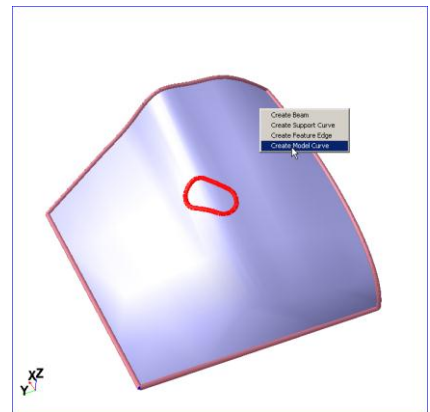
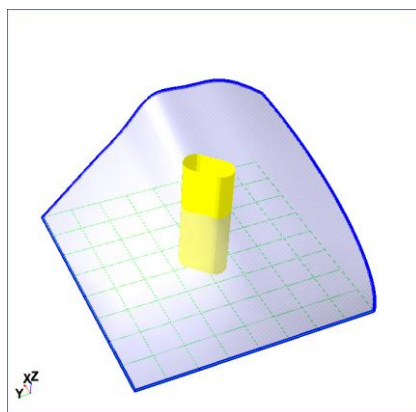
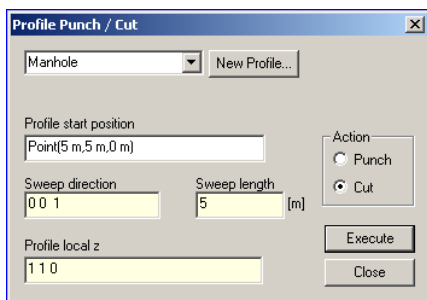
This operation is the same as *Fillet Curves*, but it works for a squared profile only.



In the example below a punch is performed (divide and trim). The solid formed from the profile and the sweep length is shown in yellow; all structural parts inside this solid are removed. Remember that the start position can not be in the same plane as the plate to punch.



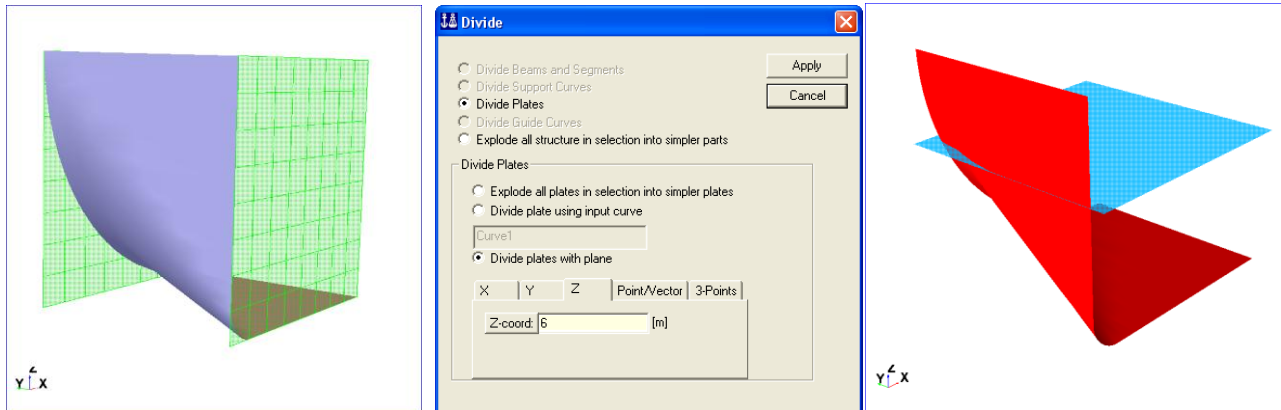
It is also possible to do punch operations on complex surfaces; below is an example of such. Different from above is that this operation will divide the plate (and stiffeners if they are inside the solid); the action type is set to *Cut*. To see the cut-lines (or topology lines) double click the plate. You can now insert model curves and use these to divide the plate.



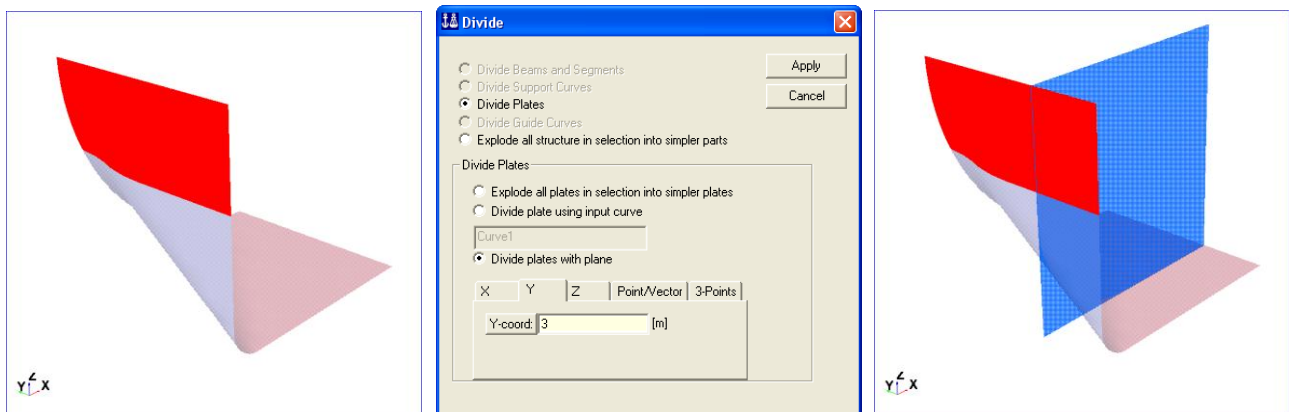
3.3.1.14 Divide using planes

You may also divide a plate (and beams) by using temporarily planes (in x, y, z or general direction). The option is available from select object(s), **RMB** and *Divide*. A typical example is shown in the following.

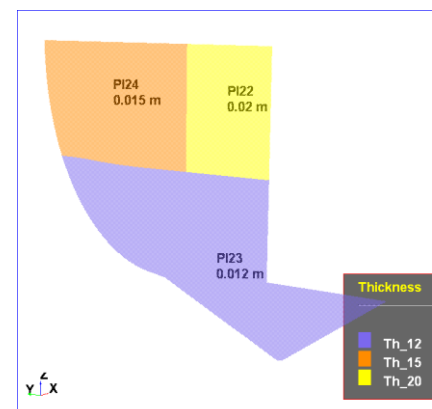
This case assumes that a part of the hull shall be divided at a given elevation and longitudinal length in order to e.g. change plate thickness. First select the plate, **RMB** and *Divide*, use option “Divide plates with plane”. The temporary plane is now shown with blue colour.



The upper part shall be divided at a longitudinal distance 3 m from the origin. Repeat the process above, but use a plane perpendicular to the x-axis.



After dividing the plates you can now assign the various plate thicknesses to the plates. The picture to the right has been made by labelling the plate thickness properties, the plate thicknesses and the plate names. Furthermore, the colour coding has been modified by using the features available from the **View Option/Color Coding**. See User manual Vol. I for further references.

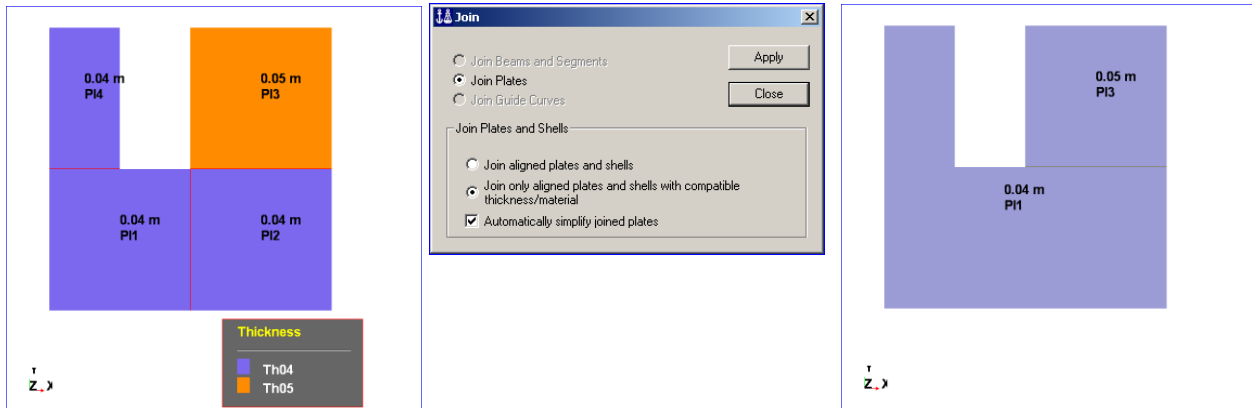


3.3.1.15 Join plates

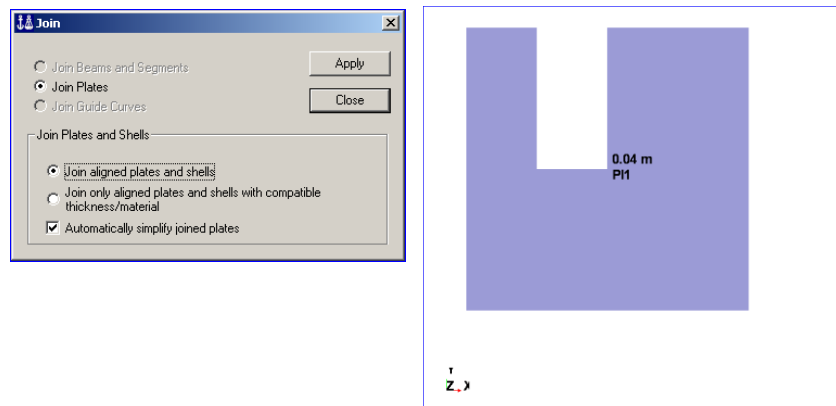
If you want to join plates to a larger plate, you can do this by selecting the plates, **RMB** and *Join*. It is required that the plates are aligned, i.e. the tangential direction is constant for all plates in the connection area.

The example below shows flat plates, but it works the same for curved surfaces.

Option 1: Join plates with same thickness only. Select all plates, **RMB** and *Join*. Use option “Join only aligned plates and shells with compatible thickness/material”. As can be seen, plate P13 is not joined with the rest since it has another thickness.

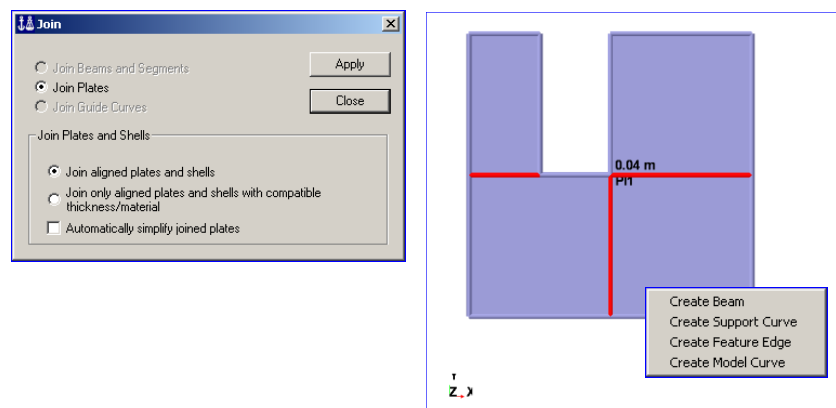


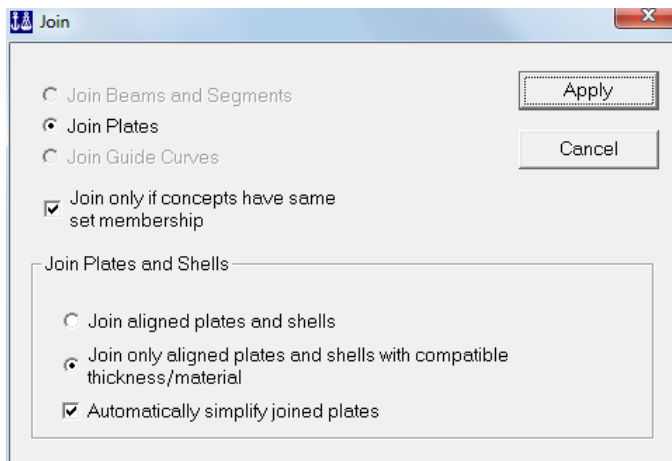
Option 2: Join all plates. Repeat the process above, but use “Join aligned plates and shells”. All plates are now joined. You should check which thickness is applied to the joined plate.



Option 3: Join all plates, but do not automatically simplify the joined plates. By using this option, the internal edges from the original plates P11 through P14 are not removed. Remember that edges like this can be used to insert a beam, boundary conditions, model curve or feature edges. As explained later in this user manual there will always be a mesh line along an edge.

The internal edges are shown to the far right (by double clicking the plate). Three of the edges are highlighted.



**Join only if concepts have same set membership**

With the option checked, plates are only joined if they are members of exactly the same sets. If P11 is member of a set and P12 is not, they will NOT be joined.

With the option unchecked, set memberships are ignored. The joined plates will inherit set memberships from either P11 or P12.

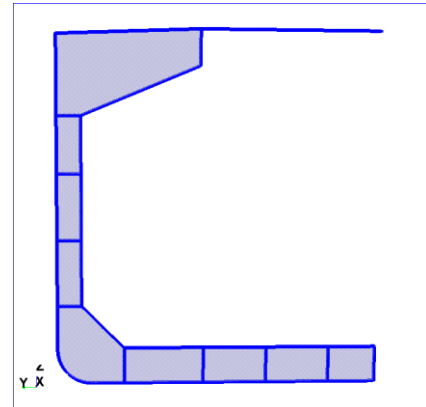
3.3.2 Use 2D structure to make 3D structures

The previous Chapter documents how to make plate and shell structures based on single or a limited number of guide curves. However, it is possible to make more complete 2D parts like for example a web-frame and use this to make a 3D structure. In most cases this is done using the sweep command.

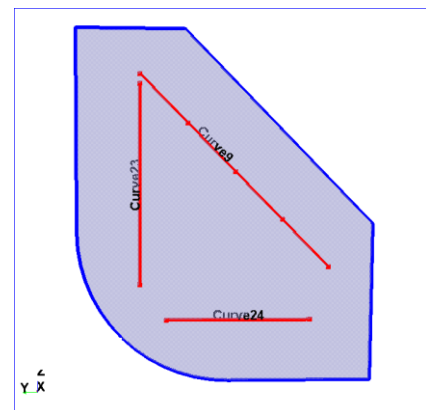
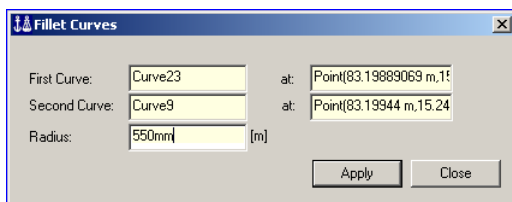
The example below shows how a typical 2D web frame (bulk ship) is created and used to make the parallel part of the vessel, typically in long and slender structure like a ship, a semi-sub, a TLP or a barge.

The starting position for this example is shown to the right. It contains guide curves and plates.

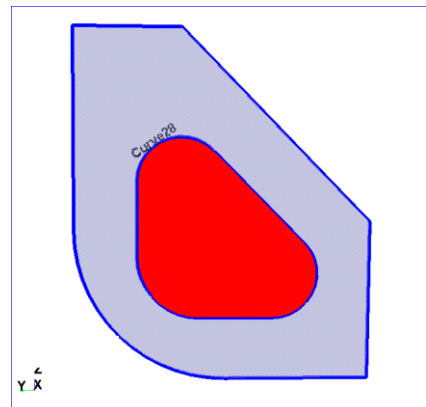
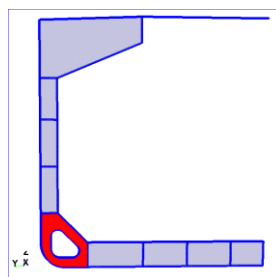
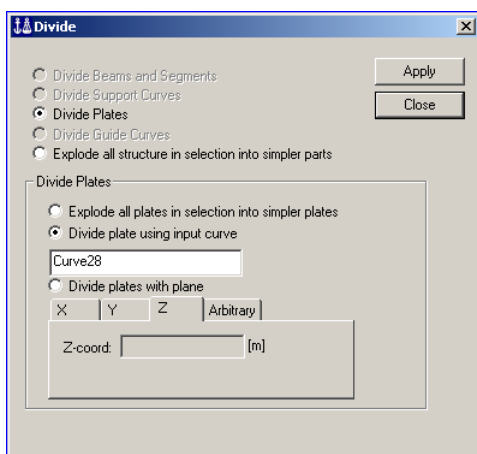
The first task is to make cut-outs using the feature for profile punch and divide using curves.



Add guide curves like shown and use these to insert fillet curves. See the previous Section “Fillet curves” on how to do this. The radius 550 mm is used for the curves as shown, for the rest 500 mm is used.

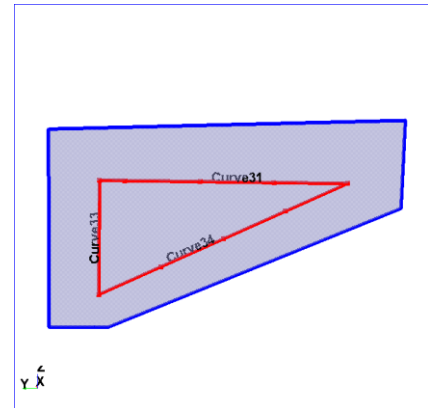
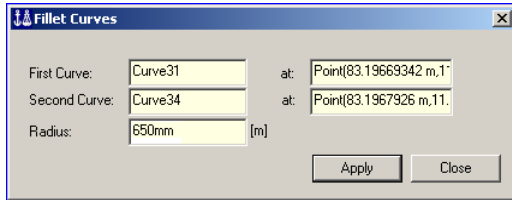


Divide the plate and use the option divide with a curve. You may now remove (delete) the plate so that you have a cut-out.

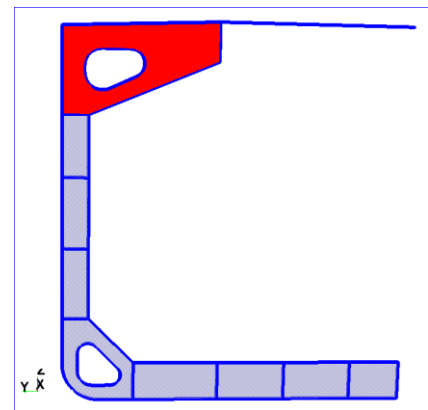


Repeat the same process at top wing by inserting guide curves and use fillet curve option.

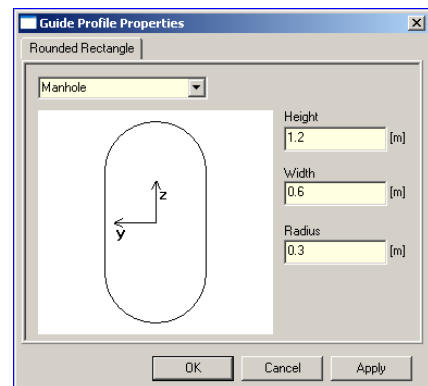
A radius of 650mm is used as shown below, for the two other corners 800mm is used.



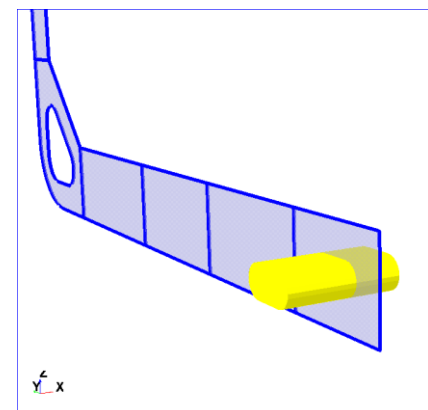
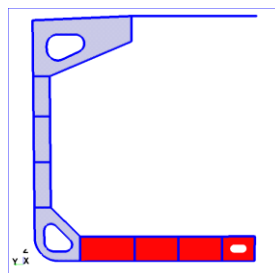
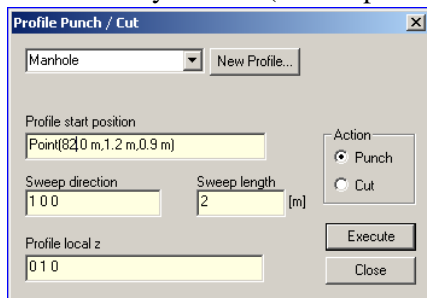
When the fillet curve operation is complete, you can divide the upper plate and remove the superfluous part.



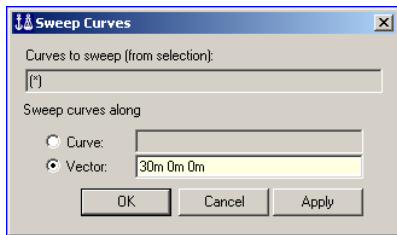
The web-frame has one man-hole – in this example it is made using the profile punching. Prior to punching a profile must be made (typically from *Insert/Profile*).



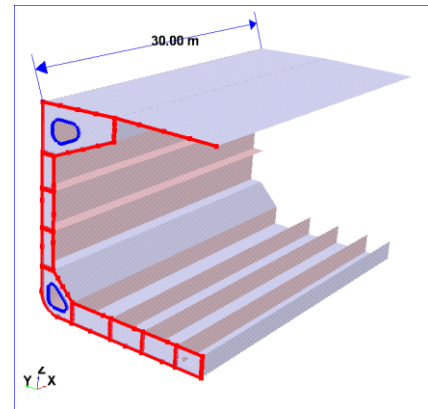
The punch operation can now be performed and the man-hole is automatically created (i.e. the punch will do both divide and delete).



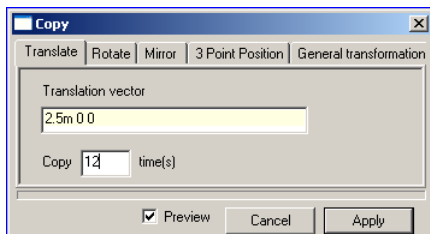
The mid-ship part can now be created using the sweep curve (also known as extrude) functionality. In this case the sweep curve uses a vector of 30m.



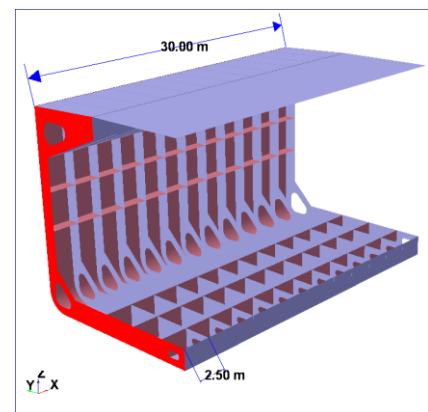
The selected curves are used in the sweep operation. Please notice that some of the plates have been removed for visibility.



The sweep curve operation will generate the structural part in the longitudinal direction. If you want to use the same web-frame at different positions you can do so by a copy operation. For this purpose it is beneficial to make a named set for the entire web-frame; you can then refer to the web-frame at a later stage.

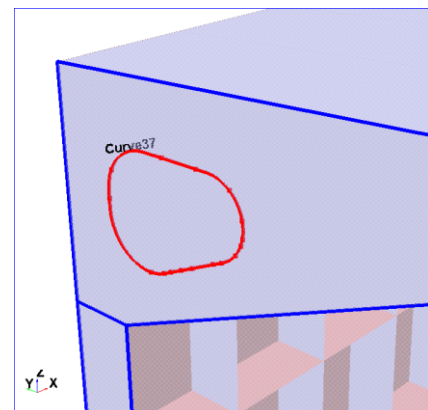


The frame spacing is 2.5m. Plates are removed for visibility.



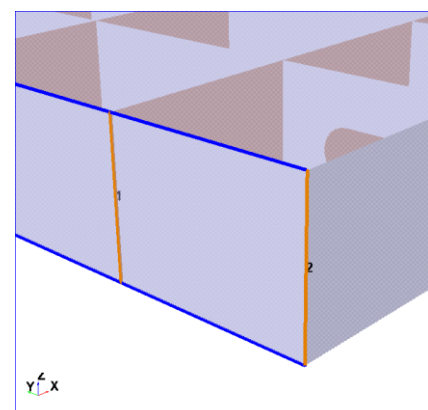
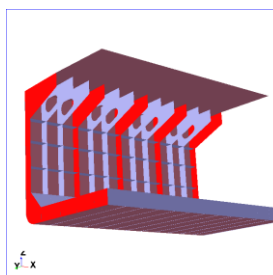
If you need to make some of the web-frames watertight (i.e. no cut-outs) you can add plates by using e.g. cover curve or skin functionality. The plates may be joined at the end.

Typically the highlighted curve to the right is used in a cover curve operation.

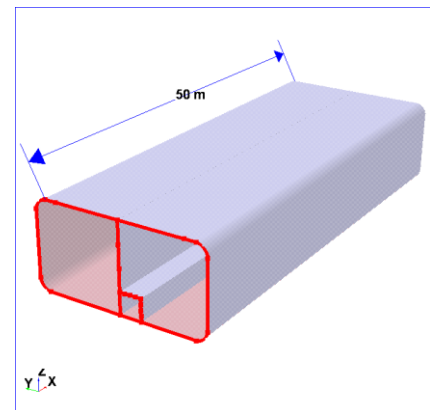
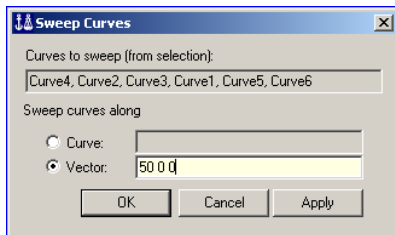


The vertical guide curves are used in a skin operation to fill the man-hole. Observe that GeniE will detect if there is any plates in-between and will only fill the open voids.

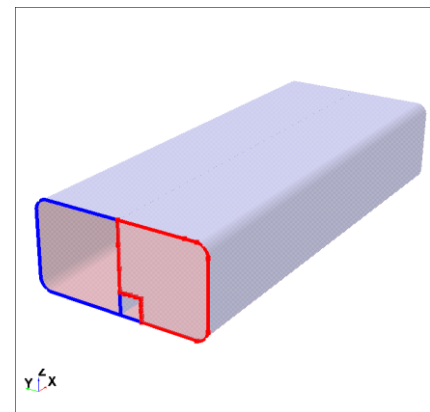
When joining the plates in the web-frame and copying the plate to selected positions, the final configuration looks like the adjacent picture.



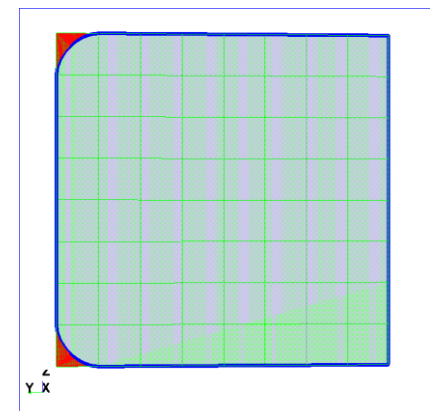
Another example may be if you want to make a longitudinal model with watertight bulkheads; in this case a part of a semi-submersible pontoon. High-lighted curves are used in a sweep operation of 50m in x-direction.



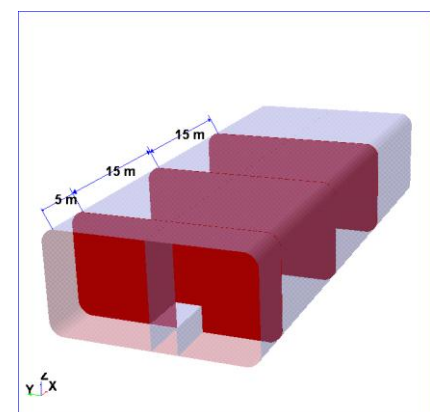
This example shows two ways of inserting watertight bulkheads; in the first case a cover curve operation is used. The watertight bulkhead on the starboard side is created by covering the void between the highlighted curves.



The watertight bulkhead on the port side is made by making a rectangular plate (same size as the guide plane), divide it by using option "Explode all plates in selection into simpler plates". To trim the web-frame to the hull the superfluous parts (high-lighted) must be deleted.



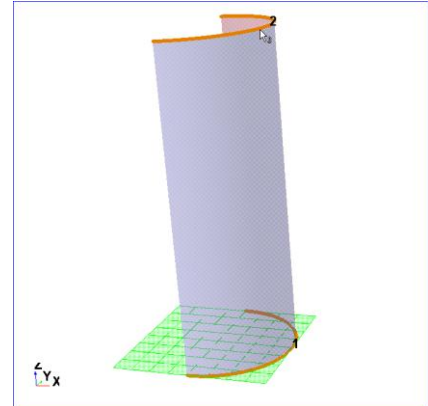
Having joined the starboard and port web-frames, it is easy to move and copy the web-frame to the selected position. In this case it is moved 5 m and copied twice with length 15 m.



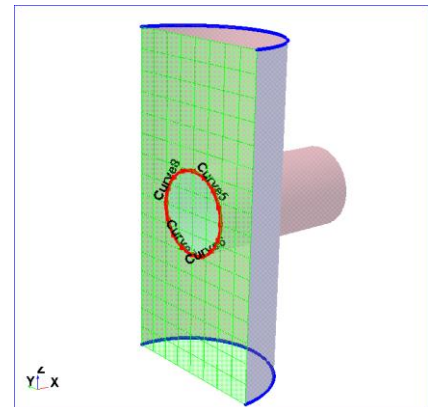
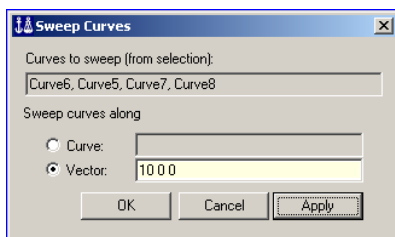
3.3.3 Make a 3D tubular joint

When making rotation symmetric structure, it is recommended to define these in segments of 45, 90 or 180 degrees. In this example, a vertical column is made by the skin operation while the incoming braces are made using the sweep curve (extrude) functionality. To trim the braces to the column and adjacent braces, divide and delete is used.

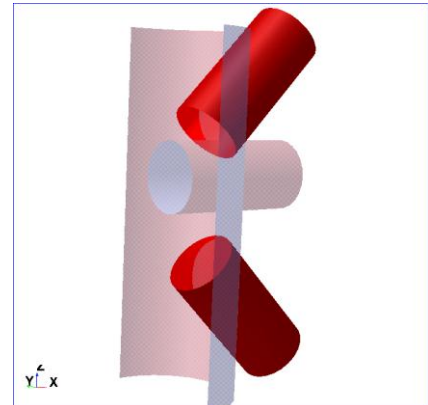
The column is made from skinning between the high-lighted curves.



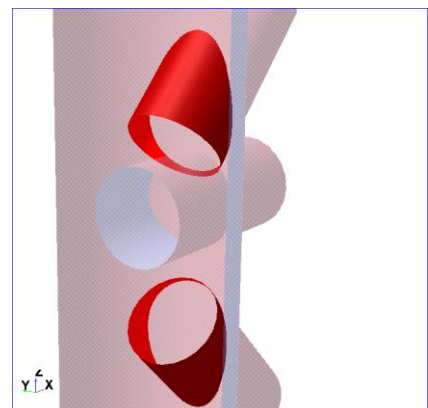
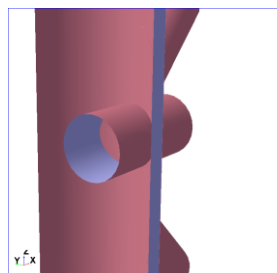
The selected curves to the right are swept 10m in x-direction to form the horizontal brace.



This example assumes that the inclined braces have the same diameter as the horizontal brace; hence the horizontal brace is copied using a rotate operation and move. The upper brace is overlapping with the horizontal brace, while there is a gap between the horizontal and lower brace. The picture to the right shows the shell configuration before divide and delete.



The inclined braces have been selected and divided using “Explode all plates in selection into simpler plates”. The high-lighted parts show which shells that need to be deleted to trim the inclined braces to the column.

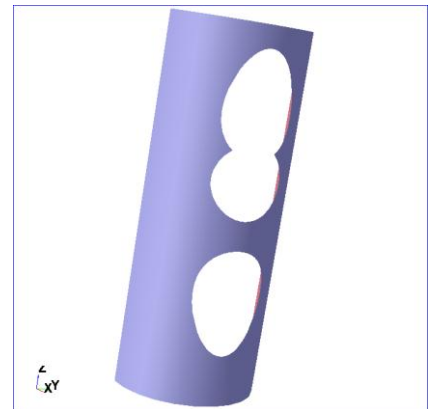
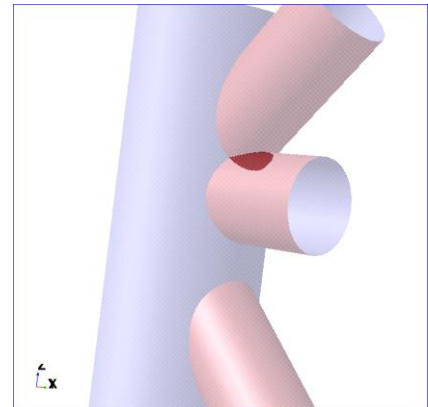
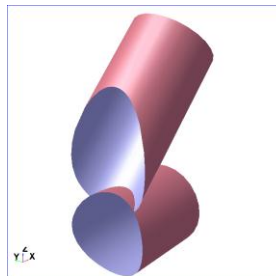


Repeat the same process for the horizontal brace. The final modelling steps are now to delete the high-lighted shells part of the inclined brace since the horizontal brace is considered a through-brace in this case.

At last you should use the command **Tools/Structure/Simplify Geometry** in order to clean up the model for unnecessary edges (see later in this user manual for more details). Alternatively, the automatic feature for topology clean-up as found under **Edit/Rules/Meshing** may be used.

The picture to the far right shows the column where the plugs have been deleted. The braces have been removed for visibility.

The upper and horizontal brace is shown to the right.

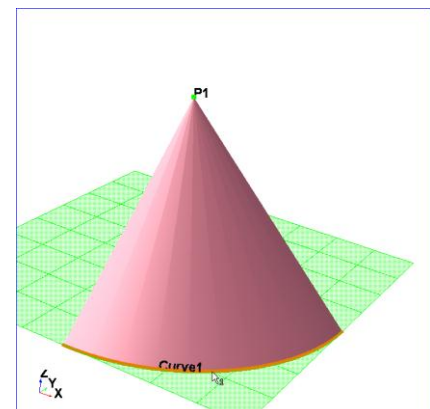
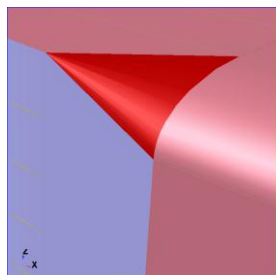


3.3.4 Make special 3D structures like cones, spheres and bulbs

Cones, or parts of cones, are often used in transition zones. This example shows how to make a cone from skinning operation. Please notice that it is also possible to do a cover operation to make a cone, but the cone shell is then not lying in the true cone surface.

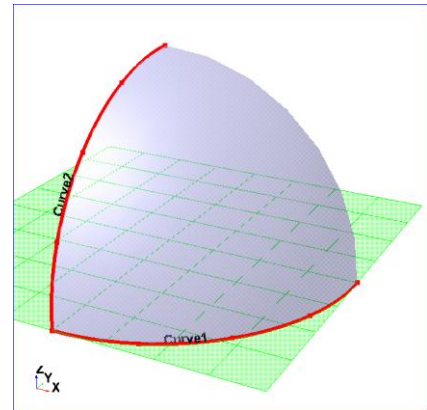
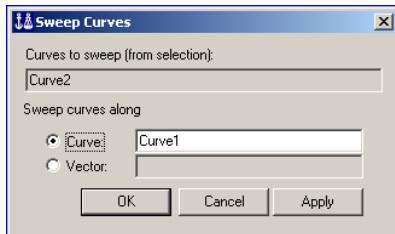
When making a cone using skinning it is necessary to refer to a guide curve and a guide point (it is not enough to use a snap point). The quart part of the cone to the right is made by selecting the curve first and then a double click on the point P1.

A typical example can be the transition between one curved shell and two plates.

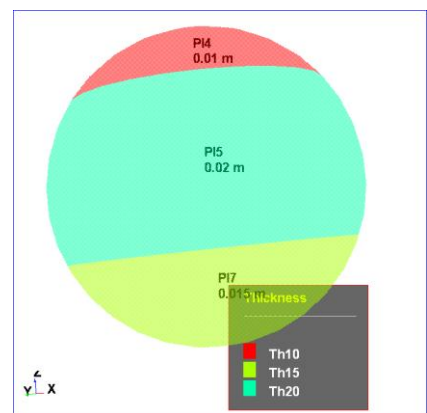


A sphere, or parts of it, can be made using both the sweep curve and cover curve option. As for the cone, it is also possible to do a cover operation to make a sphere, but the sphere shell is then not lying in the true sphere surface

The quadrant to the right has been made using sweep curves referencing curves *Curve2* and *Curve1*.



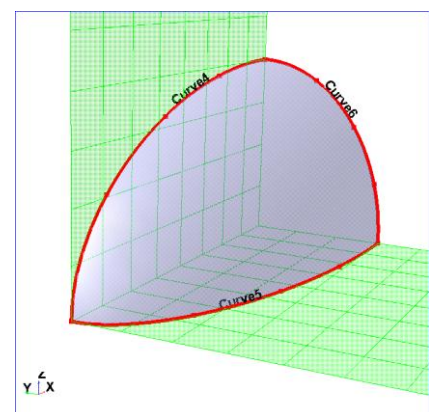
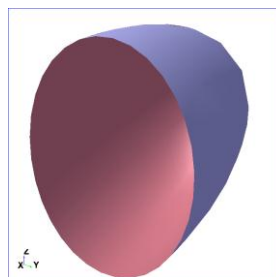
The complete sphere to the right has been created by copying the above quart part. The sphere has been divided using a temporarily snap plane at elevations $z=3\text{m}$ and $z=-2\text{m}$ in order to add a variation in shell thicknesses.



Ship bulbs may have different forms. They can be described by a set of offset curves (in this case you can do skinning between such curves) or a quarter part of it can be described using three curves.

The quarter part of the bulb to the far right has been created using a cover operation of curves *Curve4*, *Curve5* and *Curve6*.

The complete bulb has been created by copying (mirror or rotate) the quarter part.

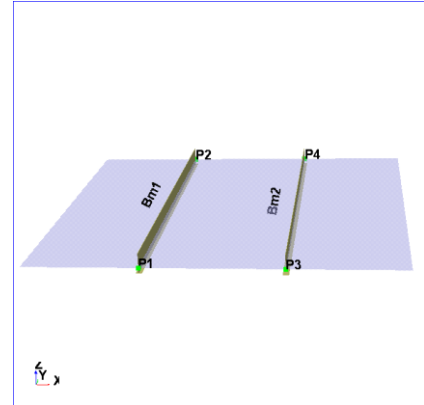
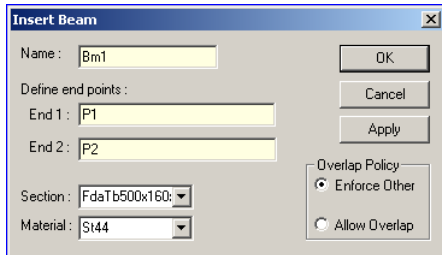


3.3.5 Add stiffeners to plates and shells

Straight or curved beams and stiffeners may be inserted in the following ways.

3.3.5.1 Straight beams between snap points

This option is normally used when modelling straight beams. If they lie in the same plane and intersect with plates or shells, GeniE will automatically ensure that there is connectivity between the beam/stiffener and plate/shell.



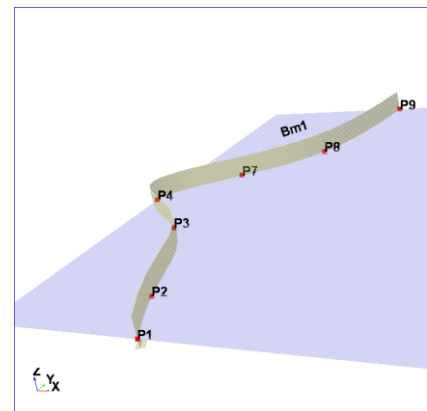
Stiffener Bm1 is defined by using **Insert/Beam/Straight Beam Dialog**. Stiffener Bm2 is created using the pulldown menu **Insert|Beam|Straight Beam** and click on the snap points P3 and P4. You can also do the same using the tool button:



3.3.5.2 Curved beams in-between snap points

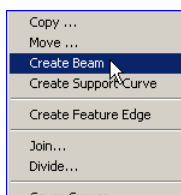
This option may be used when modelling curved beams; or in most cases stiffeners. If they lie in the same plane and intersect with plates or shells, GeniE will automatically ensure that there is connectivity between the beam/stiffener and plate/shell.

The stiffener Bm1 is inserted using the pull-down command **Insert/Beam/Curved Beam** and clicking the points P1 -> P9. Alternatively you can do this from the tool button:

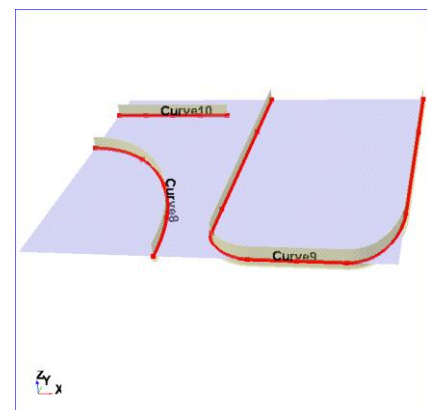


3.3.5.3 Straight or curved beams from guide curves

The stiffeners to the right are all made by creating a beam from a guide curve (Curve8, Curve9 or Curve10). Select a guide curve, **RMB** and select **Create Beam**. If they lie in the same plane and intersect with plates or shells, connectivity will automatically be made.



Curve10 will give a straight stiffener, while the other curves will give curved beams.

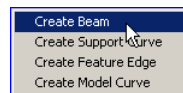
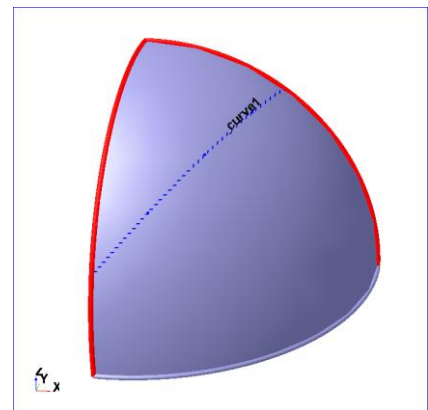
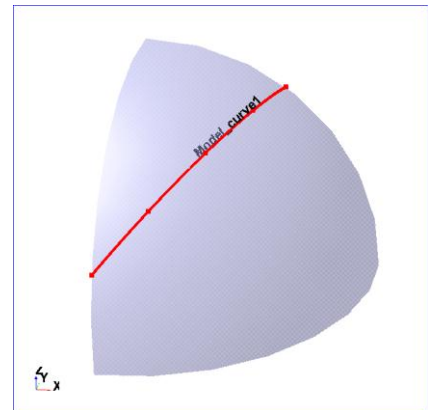


3.3.5.4 Straight or curved stiffeners from plate or shell edges

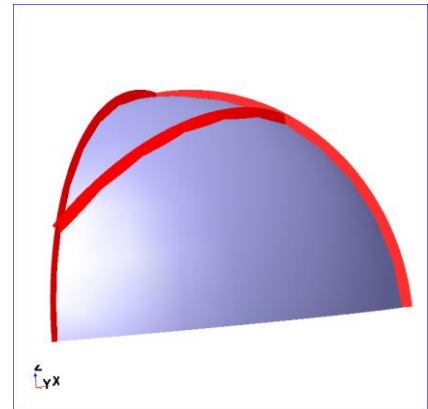
A safe way to ensure that there is full connectivity between a stiffener and a shell you can create the stiffener based on the shell edge topology. The same apply when inserting a beam from a model curve. This method can also be used for plates and stiffeners, but in this case there will normally be full connectivity since both plate and stiffener lie in the same plan.

To do this you double click a shell, select one of the plate edges, **RMB** and *Create Beam*. It is also possible to select multiple edges like in the example to the right, in this case at two of the edges. To get back to normal modelling view you double click the shell again.

This example has a model curve (*Model_curve1*) and the stiffener created from this one is also shown.



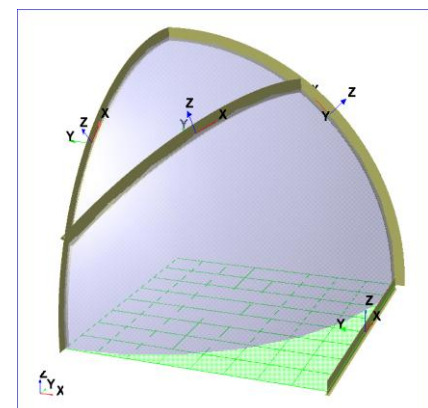
The picture to the right shows the three stiffeners created from two shell edges and one model curve.



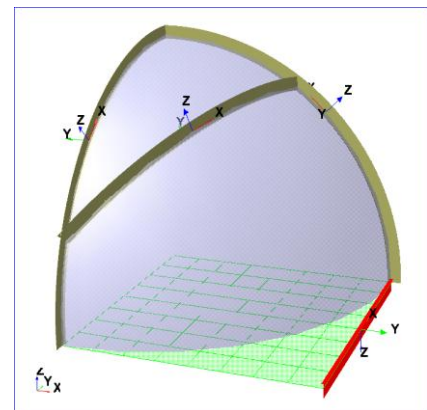
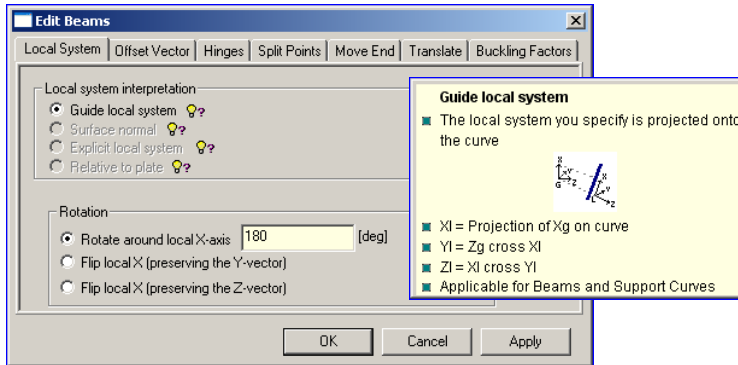
3.3.5.5 Orientation of beams and stiffeners

The program default when inserting a beam or stiffener is a local x-axis vector from first to second modelling point of the beam, while the local y-axis will sweep in the global XY-plane until the beam is vertical (then local y-axis is along global X-axis). If you want to modify this you can do it from selecting the beam, **RMB**, *Edit Beam* and choose *Local System*. Prior to such you should check the orientation from the graphics by selecting the beam, **RMB**, *Labels* and *Local Coordinate System*.

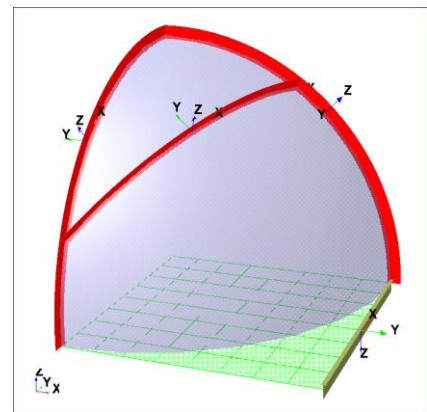
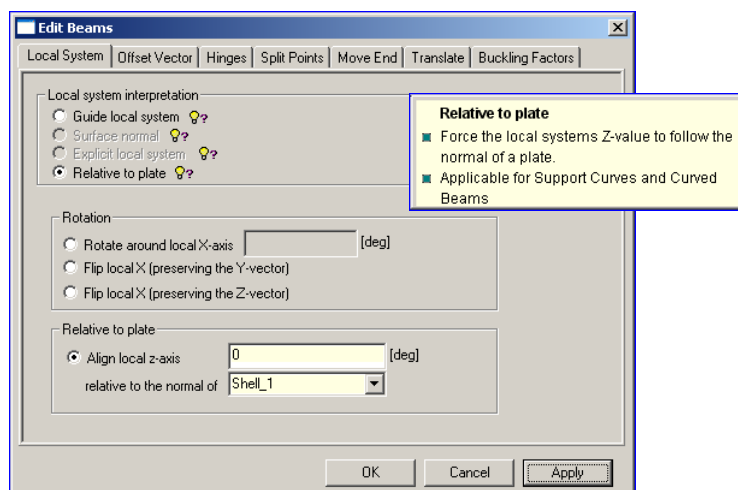
The example to the right has three curved stiffeners and one straight beam.



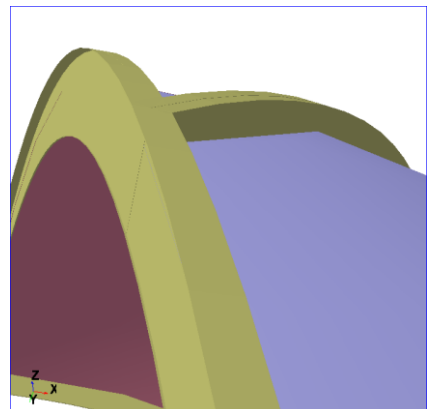
For the straight beam the local coordinate system is controlled by rotating relative to the local x-axis. It is also possible to change the direction of the local x-axis by flipping the beam and preserving the local y or z axis. In this case the beam is rotated 180 degrees as can be seen to the right.



For a curved stiffener you can do the same; in addition you can specify that the local z-axis shall always be aligned with a shell or plate. In this example all the curved stiffeners are aligned with the plate so that the z-coordinate is always perpendicular to the shell (*Shell_1*).



By using 0 degrees (as shown above) the stiffener local z-axis will be aligned and in the *same* direction as the shell normal (shell local z-axis). Typically, using 180 degrees the stiffener local z-axis will be in the opposite direction as the shell normal. An example is shown to the right where the flange is now on the other side of the shell.

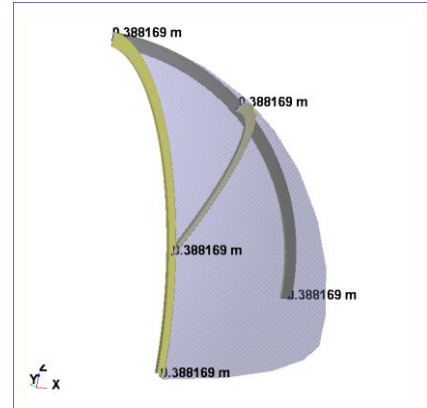
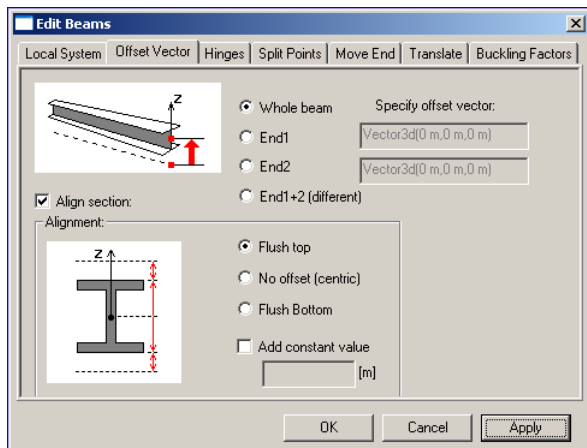


3.3.5.6 Flush stiffeners

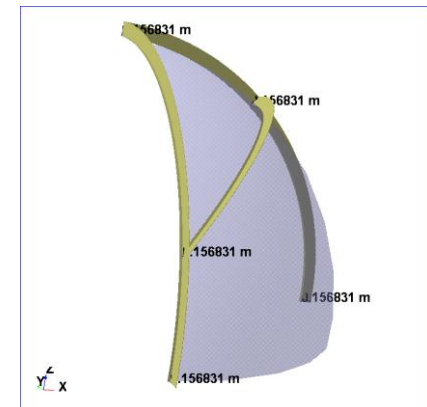
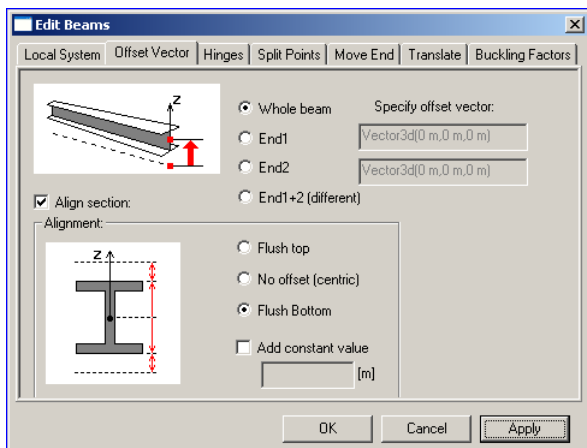
Stiffeners may be flushed automatically to plates and shells or you may specify a given eccentricity. Notice that plates and shells are modelled in their neutral axis (or *top of steel*) and flush and eccentricity parameters are applied to beams and stiffeners. For more details, see Vol. 1 of the User Manual.

To insert eccentricities like this you select the stiffener(s), **RMB**, *Edit Beam* and choose *Offset Vector*. The stiffeners you select may have different section properties; GeniE will automatically detect the offset vectors when flushing them to the shell or plate.

In the picture to the right the stiffeners have been flushed to the top of the shell; to see the eccentricity values select the stiffeners, **RMB**, *Labels* and choose *Eccentricities*.



In the example to the right, the stiffener is now aligned with the shell using option *Flush Bottom*. Since the profile used is not symmetric, the offset vectors are different from the case above.



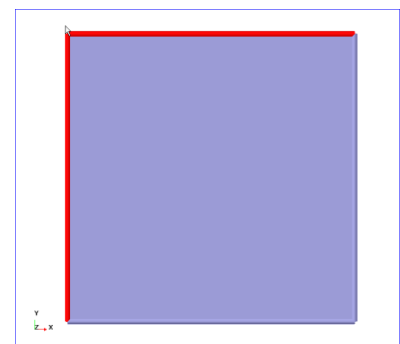
3.3.6 Topology, edges and vertices

Edges and vertices are used by GeniE internally to describe the structural parts like plates, shells or beams or when connecting geometrical entities like straight and curved parts. In the latter case edges are denoted *internal edges*. Edges and vertices are automatically created during insert (including copy) operations and they have an impact on the finite element mesh that is created. It is therefore important to understand and control the edges and vertices since they have an impact on the quality of the finite element mesh.

The topology describes the connectivity between structural parts, i.e. how many parts and in which directions are they connected to a given edge. When you insert, copy, move or delete parts the topology is always updated, but un-necessary topology is not automatically removed by program default. The topology has also an impact on the finite element mesh and you can remove the un-necessary topology either manually or per default when you create a finite element mesh.

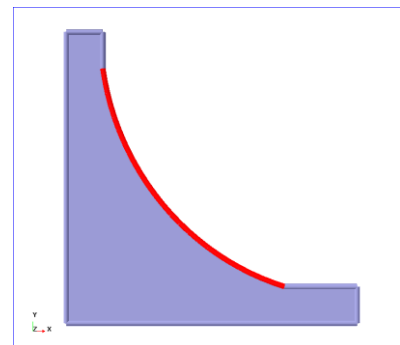
Please consult the Chapter *Make and control the finite element mesh* to learn how to control the impact of topology, vertices and edges to the finite element mesh. In the following, some examples are given.

In the example to the right, the highlighted lines are edges to describe two of the boundaries of the plate. The common point between the edges is called the vertex. The plate shown has thus 4 edges and 4 vertices as minimum, but may have more.

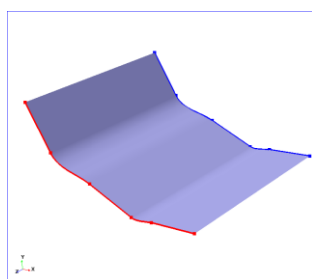



Similarly, for a straight and regular beam the edge is used to describe the beam and there are thus two vertices. A segmented beam has one or more edge per segment and corresponding number of vertices.

The example to the right shows a plate that has been made up of straight and curved lines. The highlighted part shows the edge describing the curved boundary of the plate and it is connected to the straight lines with two vertices at its end. This plate has 7 edges and 7 vertices.



The shell as shown below has been created by a cover operation between two identical complex curves (built up by spline and straight parts). When using cover, skinning or curve-net interpolation the shell will consist of different geometric primitives; flat and curved parts. The geometric primitives are connected by internal edges as shown to the right. One internal edge and two edges are highlighted. This shell has thus 16 edges in total (4 internal edges) and 12 vertices (6 on each side).




Poly Curve
✕

Name:

Fit Curve To View

OK

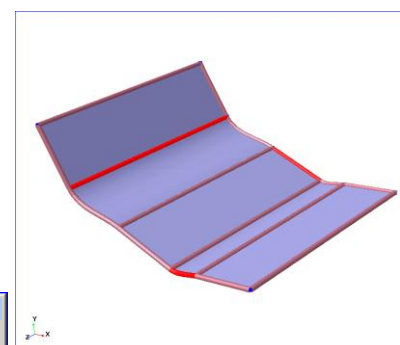
Curve definition OK

Auto Curve Type

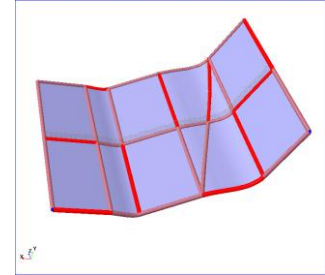
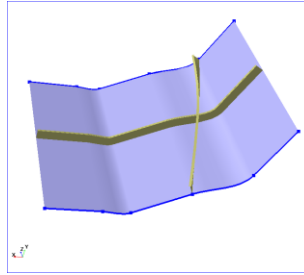
Cancel

☐ Label curve points

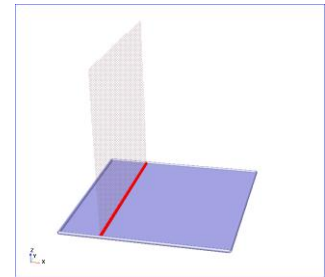
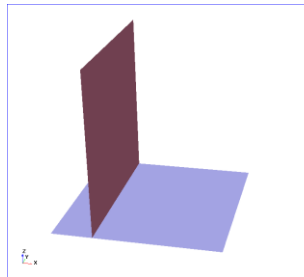
	X [m]	Y [m]	Z [m]	Curve Type
1	0	5	0	Straight
2	1.25	2.5	0	Spline
3	3.75	1.25	0	Straight
4	6.25	0	0	Spline
5	7.5	0	0	Straight
6	10	0	0	Spline



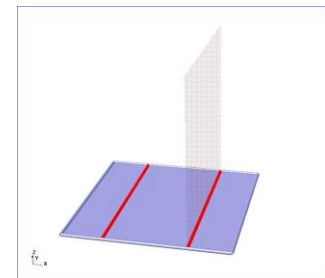
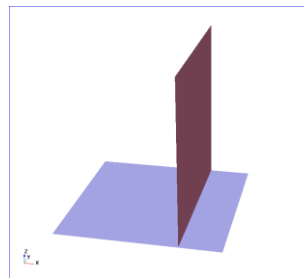
This example includes two stiffeners as shown and some of the edges (including internal edges) are highlighted to indicate typical edges. The number of edges now increases to 30, while number of vertices becomes 19.



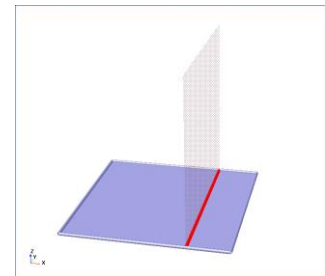
The example to the right shows the edge that is common (the topology line) for the two connected plates. There are similar topology lines when plates and stiffeners intersect. When two beams intersect (they cross each other) a topology point is established.



When copying the vertical plate to a new position, a new topology line is established and the old one is kept.



There are two ways of removing un-necessary topology lines (or points); manual or automatic during meshing operations. The **Tools/Structure/Geometry/Simplify Topology** removes topology not used to describe the connectivity between objects. You may also specify that the simplify topology shall be an automatic part of meshing operations. In this case the topology line used to describe the connectivity between the horizontal plate and the vertical plate prior to copy is removed.



3.3.7 Verify the concept model

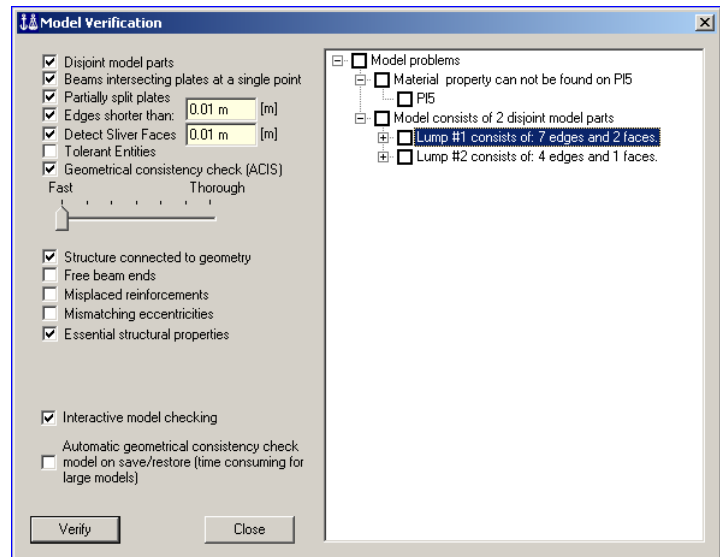
It is possible to verify the quality of the model prior and after making a finite element model. In the first case focus is on the concept model while the latter case helps you to identify finite elements that might lead to problems during the analysis itself or results with low quality. This Section shows how to verify your concept model.

The model verification is done by using the pull-down menu **Tools/Structure/Verify**.

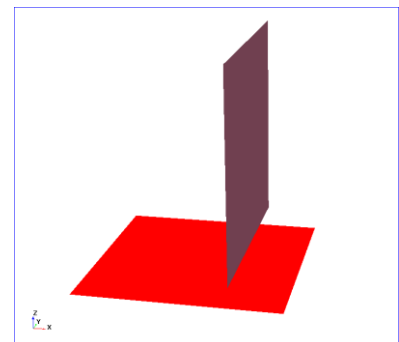
The picture to the right shows the system default settings. If you change these, they are persistent from project to project; i.e. the settings are kept until you change them.

Each of these options is described in the following.

If there are problems in your model, the model verification tool will report problems in the browser area of the dialogue. In this case there are two problems; material missing and 2 disjoint parts in the model.



When clicking (LMB) on the various problem descriptions, the relevant structural parts will be highlighted in the graphical window; in this case one of the disjoint parts have been selected from the model verification browser. When using this technique you can e.g. decide to view only the parts causing problems or you can save them as a named set for easy access to the problem areas at a later stage.



Disjoint model parts

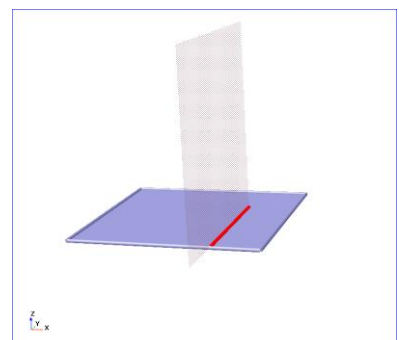
This option checks if there are model parts that are not connected to others. You may create a finite element mesh, but the analysis will fail because there is more than one model.

Beams intersecting plates at a single point

A beam penetrates a plate or shell using a topology point to describe the connection. This will lead to problems during meshing. To come around this problem you need to add edges to the topology either by splitting the plate or to insert so-called feature edges (see further in the Chapter *Make and control the finite element mesh*).

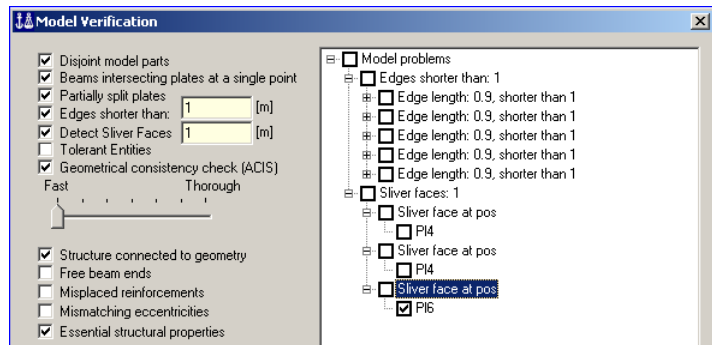
Partially split plates

This is reported when you have a plate that partially split another plate, i.e. does not cross two edges. However, a finite element mesh will be created and analysis can be carried out.



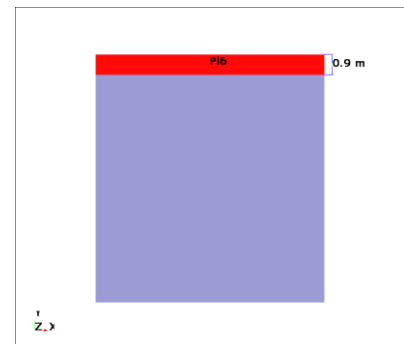
Short edges and sliver faces

In this example a plate with width 0.9 m has been added to an existing plate. By using a criteria 1.0m when checking for short edges and sliver faces GeniE will detect and report those less than the specified criteria. In this case the sliver face for plate P16 is selected from the browser and graphically highlighted. Short edges can be detected using the same approach.



Short edges and sliver faces may have a significant impact on the finite element mesh and should be avoided as far as possible.

In real cases you should have far less values than 1.0 m as criteria when checking for short edges and sliver faces; the value 1.0 m was chosen because of visualisation in this user manual.



Tolerant entities

The geometry modelling in GeniE is using a 3rd party software tool called Acis which requires its input to be highly accurate, to the order of 1e-6. This requirement may in some cases be difficult to satisfy; for this purpose there exist an Acis feature called *Tolerant entities*.

Example: If you have, for some reason, a very short edge, or you insert a beam that almost intersects existing structure, Acis may replace the short edge with a tolerant vertex. This vertex will in effect represent two model positions at once. This vertex has a different model tolerance than the rest of the model. Points that lie within e.g. 1e-3 may be considered to be equal to the position of the tolerant vertex. Tolerant edges can also be used to replace sliver faces in the model (automatically done by Acis).

In some cases the tolerant entities may introduce model problems, so it may be relevant to check for the existence of them.

Geometrical consistency check

This is an internal sanity check of the geometrical Acis model. It checks if intersections between plates have been calculated correctly, if the parameter ranges of edges corresponds to the underlying parameter range for the curves and so on. The geometrical consistency check has several hundred failure criteria to use in order to detect what makes a model invalid. Some of these criteria are heavier to compute than others. If you select *Thorough* checking, you will test against all failure criteria. This can be time consuming for large models. If you choose *Fast*, you will only get the most obvious geometrical errors. These errors usually come up because of internal computation errors in Acis, but they may also be triggered by inexact modelling by the user (very short edges, sliver faces etc.).

Structure connected to geometry

Check if the concept model is somehow out of sync with the geometrical model. A common example of this can be support points modelled in loose air; Acis is unable to represent free vertices in a model, a vertex must be connected to an edge or a face.

Free beam ends

This option will check if there are beam ends not connected to other structure. A typical example may be secondary beams not properly connected to the primary structure.

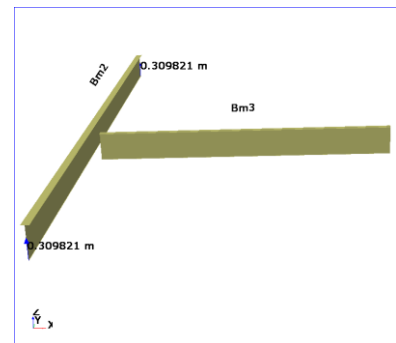
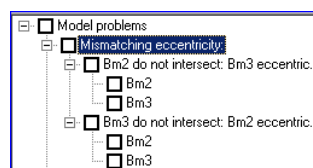
Misplaced reinforcements

Check if a reinforcement property is assigned to a segment that is not the nearest neighbour to a joint. It is required that a joint concept has been created (typically from *Insert/Joint/Joint Dialog*).

Mismatching eccentricities

This check will detect if you have intersecting beams that don't intersect eccentric; typically when one out of two beams have eccentricity.

In this example BM2 has an eccentricity vector (0.309821 m) while Bm3 has no eccentricity. This check reports that there is mismatch between Bm3 and Bm2.



Essential structural properties

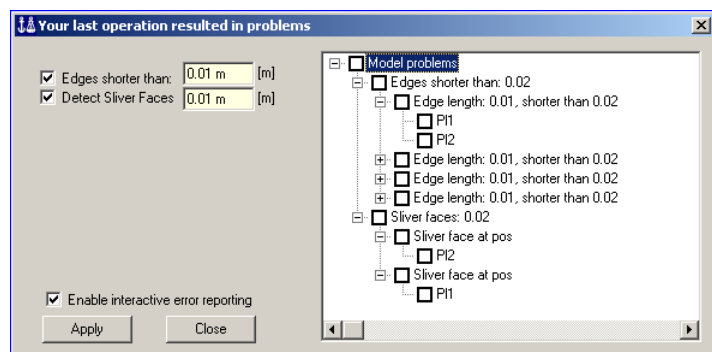
This check will list structural parts that have not been assigned section property, thickness property or material property. If some of these are missing, the structural analysis will fail.

Interactive model checking

This option will check your model every time you do an update to your model. You should remember to set the search criteria for each project.

It is recommended that you use this feature – it is program default, but with no criteria specified.

Typically if you copy a plate 0.005m (or a small value) you may create unwanted short edges and vertices. In this case you will be prompted with the following message:



Automatic geometrical consistency check

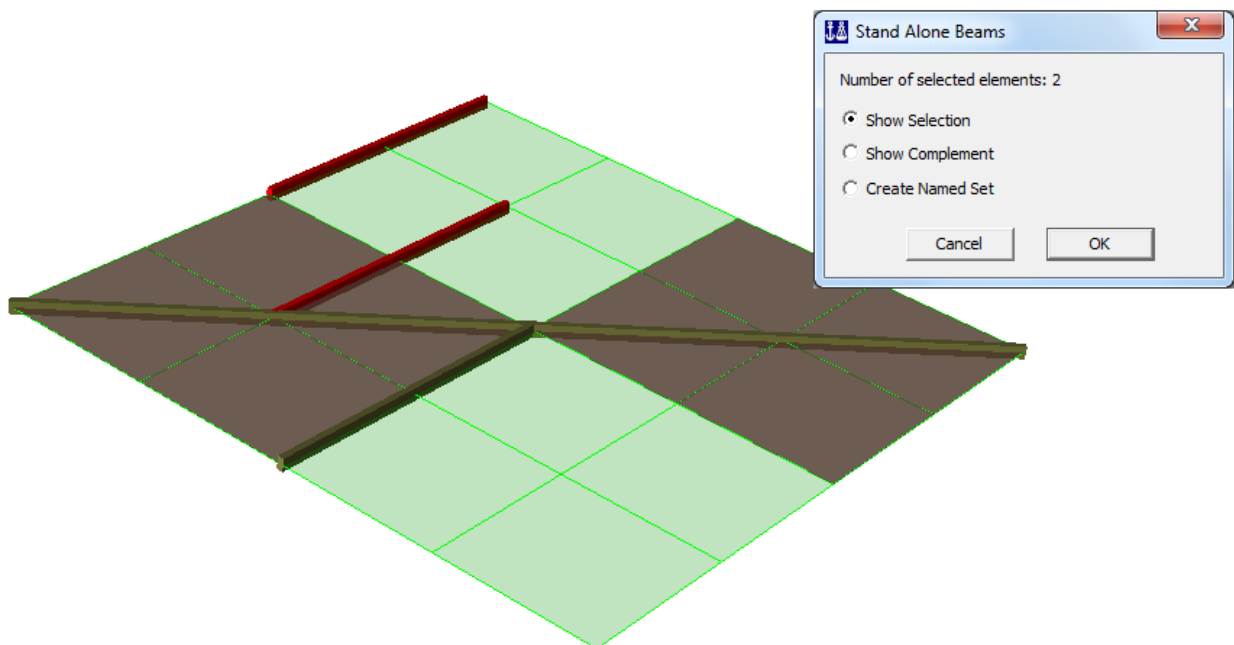
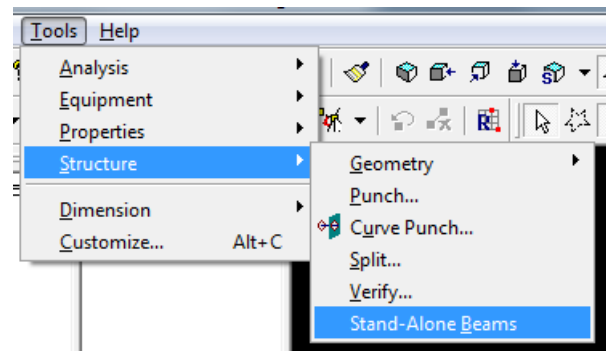
When this option is activated, it will perform a geometrical consistency check each time you save your model.

Free plate edges

It is also possible to visualize free plate edges (plate edges not connected to other objects) by selecting the plate, **RMB** and select *Labels/Geometry/Free Plate Edges*. The free edges are now shown with a yellow line.

3.3.8 Stand-Alone Beams

In order to find “stand alone beams” in the model, select the menu item **Tools/Structure/Stand-Alone Beams**



All beams that are not fully attached to a shell are classified as “Stand-Alone”. In the illustration above, the red beams are “stand-alone”.

It is possible to show the selection only, to hide the selection and show the complement and also to create a named set and add the beams into it.

This tool can be useful in cases where you observe errors due to instabilities introduced by stand-alone beams in the model (e.g., in compartment creation).

4. MASSES, LOADS AND COMPARTMENTS

In GeniE you can do design load based analysis, prepare models for use in a direct analysis (i.e. interaction with hydrodynamic analysis) or perform eigenvalue analysis. Each of these analysis types use mass and loads differently.

Some characteristics for design load based analysis:

- Apply explicit loads directly to beams, stiffeners, plates or shells. Examples of explicit loads are point loads, line loads and surface loads.
- Place equipments. This will generate line loads or inertia loads accounting for the equipment's centre of gravity, footprints and which beams are designated to carry the equipment mass.
- Inertia loads. These are loads that are computed by the program multiplying the mass (structure, point mass, equipment mass) with a gravity field (constant or varying like e.g. a centripetal acceleration field).
- The compartment filling is computed by GeniE to pressure loads. You may also define explicit pressure loads inside a compartment.
- In case you are using the Nauticus Hull rules for CSR Bulk, the load conditions (explicit pressure loads) are automatically defined for you based on your compartment definitions.

For models to be used in a hydrodynamic analysis (direct analysis):

- The model normally consists of mass (structure, point mass and equipment), compartment content (defining which compartments to be part of hydrodynamic analysis) and explicit loads.
- It is also common to make different hydrodynamic models like the panel model (defining outer wetted surface), the structure model (defining typically the compartments and their contents) and the mass model.
- The hydrodynamic analysis will provide surface loads and accelerations in addition to the explicit loads already part of the model. These are automatically used during the structural analysis; hence all effects can be considered during post-processing, code checking stage or during fatigue analysis.

Eigenvalue analyses are based on stiffness and the mass. It is possible to represent equipments as line loads or mass; this means that for an eigenvalue analysis the equipments must be considered as a contribution to the total mass.

This Section shows how to apply explicit loads to beams, stiffeners, plates and shells. Furthermore you will also learn how to create compartments and fill them with liquid or solid content and how to compute forces based on the content. Finally, this Chapter discusses how to apply pressures inside the compartments.

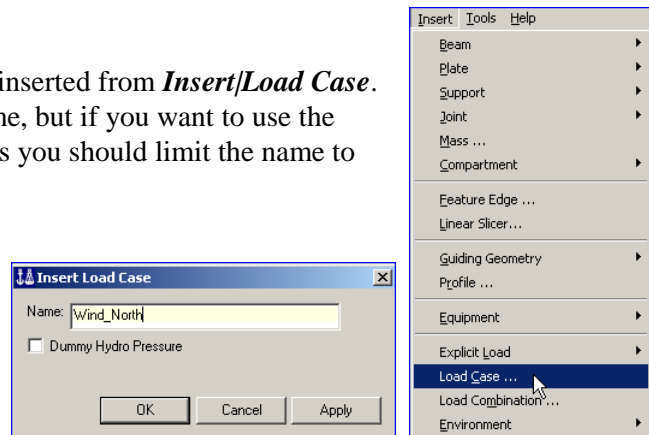
Volume 1 of the User Manual explains how to deal with equipment modelling, mass modelling and how to work with different acceleration fields.

4.1 Load cases and load combinations

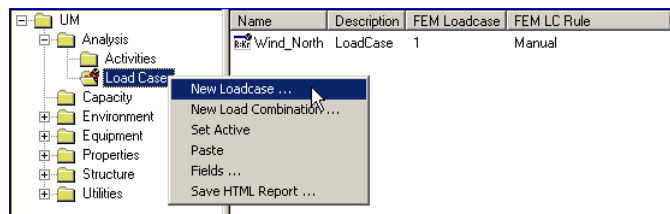
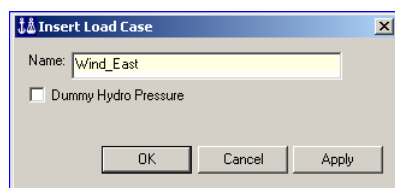
Load cases and load combinations can be created at any time. Before loads can be generated it is necessary that a load case is selected by setting it to “current”.

You can have different analysis activities in GeniE where you have different load cases. If you want to have different analysis activities the loads defined in the Load Case folder will apply to all analysis activities, while the loads defined in the analysis activity folder apply to the particular activity in question. It is possible to deselect a global load case in an analysis activity.

Load case to be used by all analysis activities can be inserted from *Insert/Load Case*. There is no limitation to the length of a load case name, but if you want to use the finite element model in other Sesam program modules you should limit the name to eight characters.

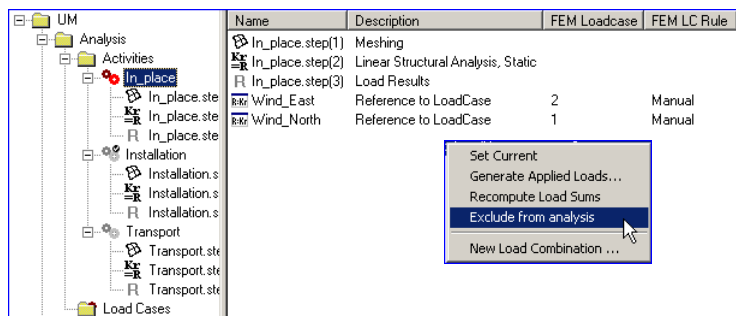


You can also define a load case directly from the browser. Click **RMB** on the Load Case folder and select *New Loadcase*.



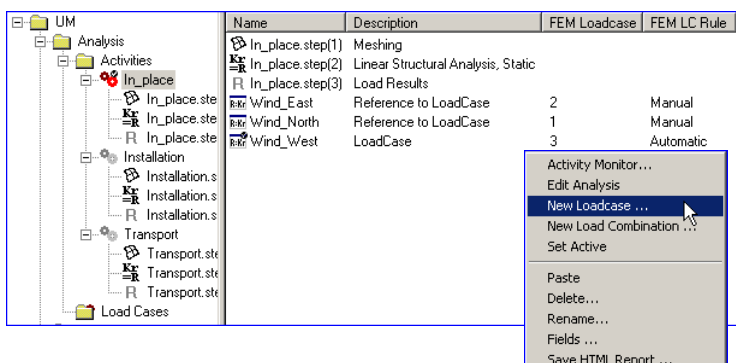
If you have specified several analysis activities, your browser will look like this before defining load cases relevant for the particular analysis activities. As can be seen the two global loadcases are part of this analysis activity by a reference to global load case.

You may exclude global load cases by selecting it, **RMB** and *Exclude from analysis*.



Load cases for a particular analysis activity are created by selecting the analysis activity, **RMB** and *New Loadcase*.

This example has thus two global loadcases to be used by all analysis activities and one load case for this analysis activity only.

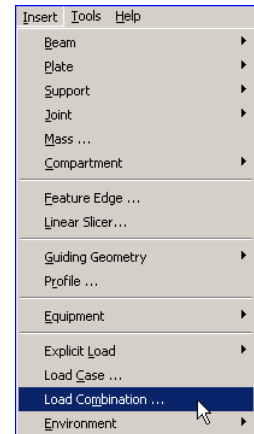
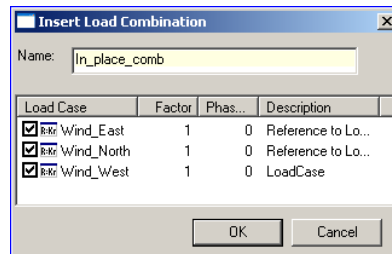


A load combination is built up from load cases or other load combinations. Each contribution to the load combination may be scaled with a factor.

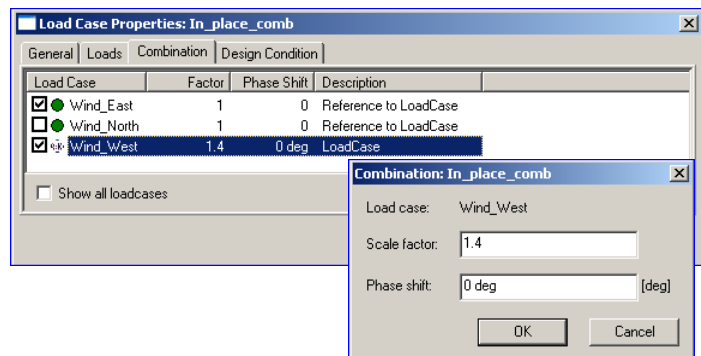
A load combination may be created from **Insert/Load Combination**. This will create a load combination including all load cases (and combinations) that are part of the analysis activity that is set to active. The scaling factor for all load cases is set to 1.0.

In this case the analysis activity “In_Place” is set to active.

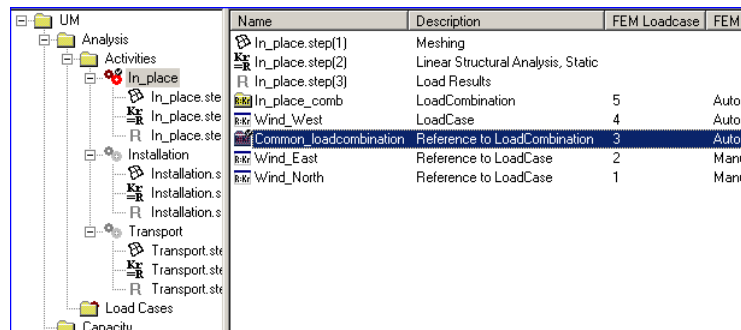
A load combination may be modified by selecting it, **RMB** and select *Properties*.



Double click one of the load cases that are part of the load combination to change the load factor or to remove the load case from the load combination. In this case “Wind_West” has a factor of 1.4, while “Wind_North” is checked off.



You may also select the load cases you want to use in a load combination, **RMB** and select *New Load Combination*. If you do this in the Load Case folder, such load combinations become part of all analysis activities unless you de-select them. The example to the right shows two load combinations of which one has been created in the Load Case folder.



It is not recommended to have a mix of load combinations created both in the Load Case and analysis activity folders. For wave load analysis (typically jacket or jack-up) all load combinations must be defined in the analysis activity folder.

GeniE has a default numbering system for load cases on the finite element model. Furthermore, the load combinations are not part of the structural analysis; the load effects are done by scaling the results from load cases. It is advised that you use these default rules unless

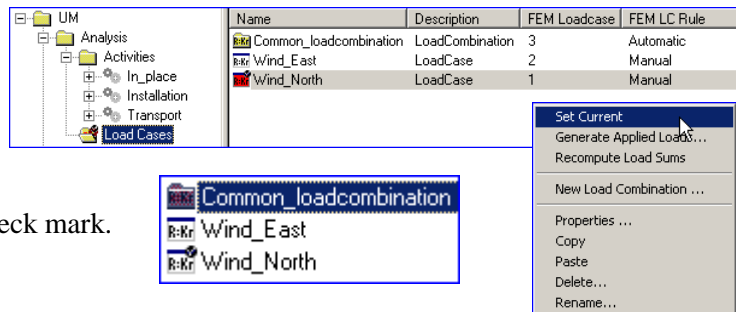
- You want to control the load case numbering on the FEM file in conjunction with load assembly in a superelement analysis
- You want to look at the loads and the reaction forces after a structural analysis from the Sestra listing file in stead on a report created by GeniE.
- You want to run a pile/soil analysis; this analysis requires that all loads are computed.

The Section *Run Analysis* explains how to change the default parameters for load cases.

4.2 Explicit loads on beams and stiffeners

It is possible to apply point loads, line loads, temperature loads and prescribed displacements to a beam. These options are described in the following. Each Chapter will describe how to model, how to change and how to document a load.

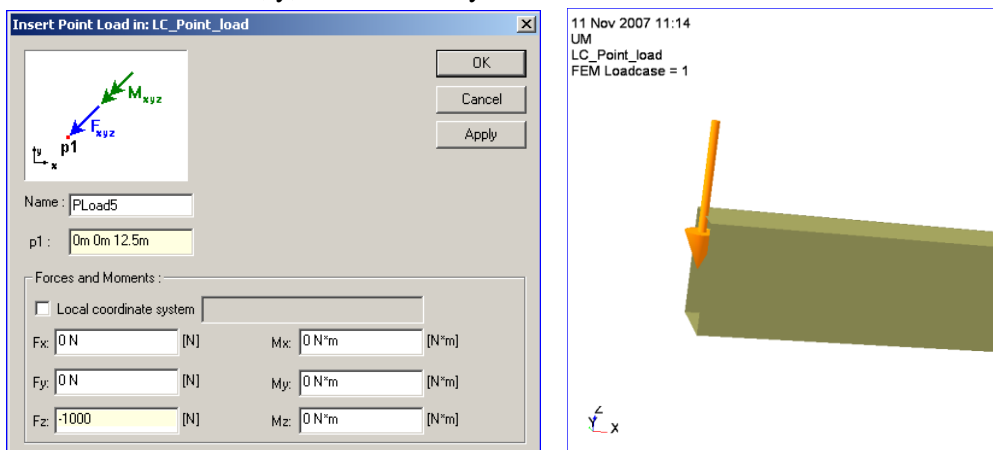
Before a load can be applied it is necessary to set a load case to active. Select the relevant load case, **RMB** and *Set Current*.



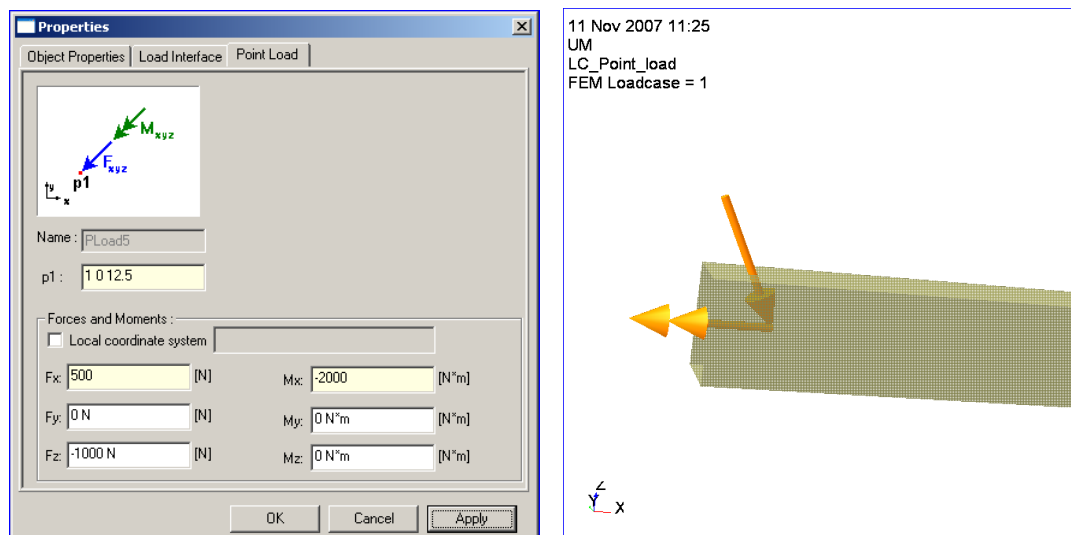
The current load case has an identification check mark.

4.2.1 Point loads on beams

Point loads can be inserted along any position of a beam or stiffener; it is not required to have a snap point or other physical connections with other object. Point loads are defined from **Insert/Explicit Load/Point Load**. The example below shows a point load of -1000N in z-direction applied at position (0m, 0m, 12.5m). Point loads may be inserted by specifying components (Fx, Fy, Fz, Mx, My, Mz) in global x,y,z directions or relative to a local x, y, z coordinate system.

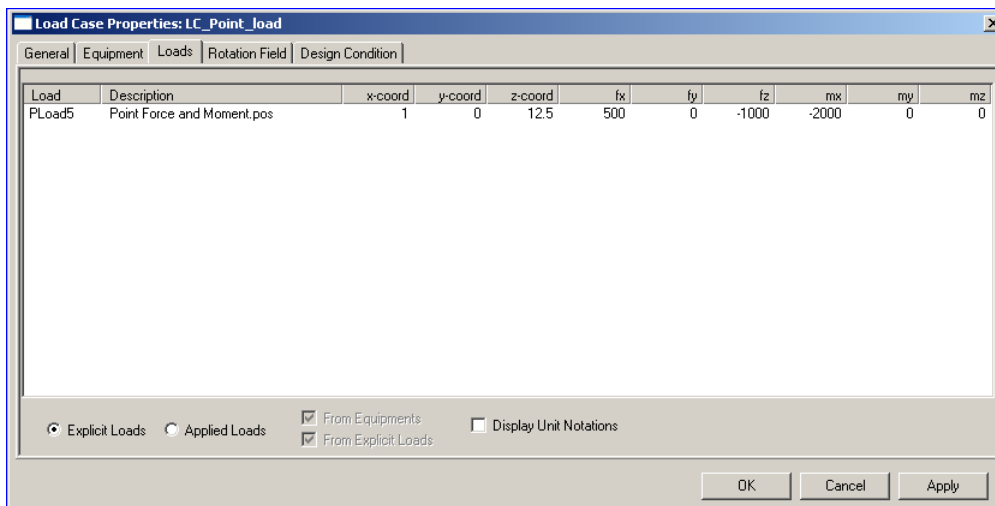
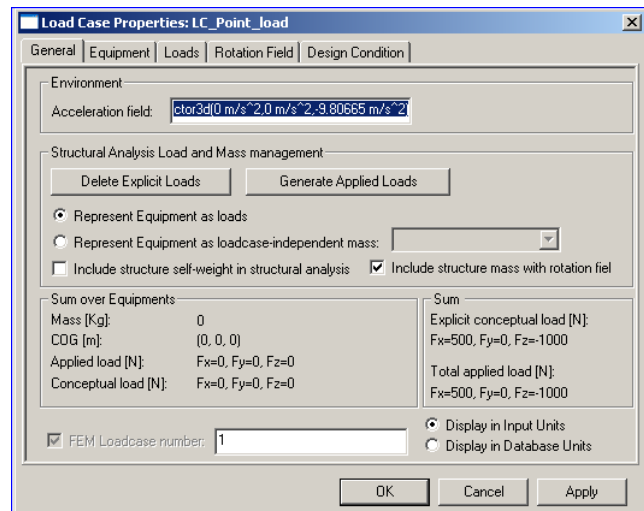


To modify a point load, graphically select the load, **RMB** and *Properties*. You may also move or copy the point load to other positions(s) from graphically select. In the example below, the load intensity has been modified as well as moved to a new position.



The point load can be documented in several ways.

- Graphic as shown on previous page
- From a report generated by GeniE; see the Section Pictures and reports for further details.
- Select the loadcase, RMB and Properties. The explicit loads can also be documented from the folder tab Loads. The example below shows both options for the point load applied in load case LC_Point_Load.

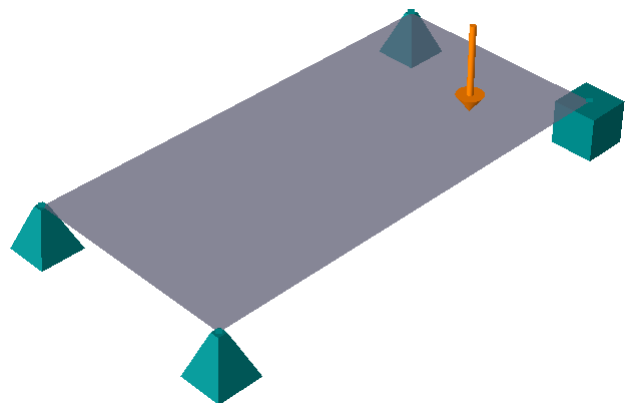


4.2.2 Point loads on plates

A point load can be applied anywhere on a plate. You make a point load on a plate in the same way that you make a point load on a beam, explained in the previous paragraph.

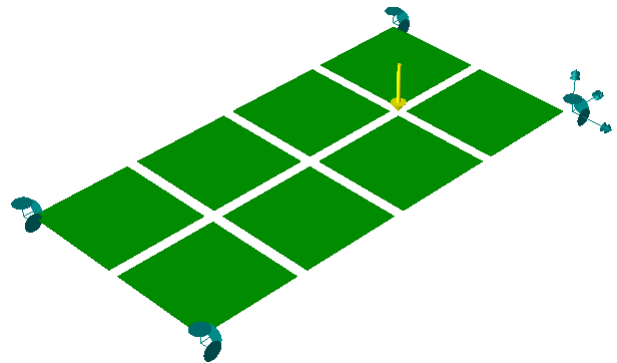
During meshing the point load will be moved to the nearest FEM node.

A point load is placed on a plate, the illustration to the right shows the model in “Default display”.



After meshing, when looking at the model in the “Mesh – All” view, it can be observed that the point load has been moved to the nearest node in the mesh.

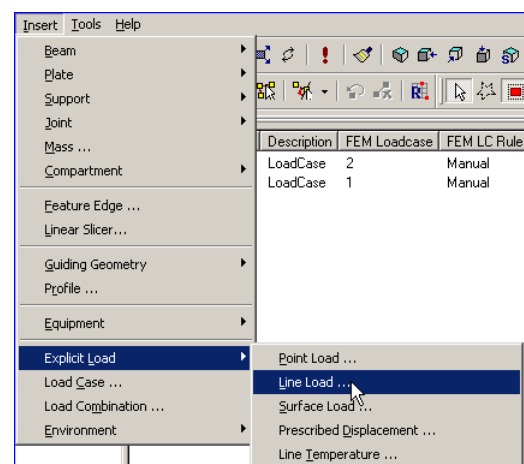
If you want to avoid the point load being moved, you can insert feature edges to change the mesh or make the mesh finer.



4.2.3 Line loads on beams

A line loads can be applied to the entire - or parts of - beam or stiffener. The load can consist of the load attributes F_x , F_y and F_z and they can be constant or linearly varying. Furthermore, the line loads may be defined as own objects or they may belong to a beam. In the first case, applied loads are only generated when a beam intersects a line load object. Both options are explained in the following.

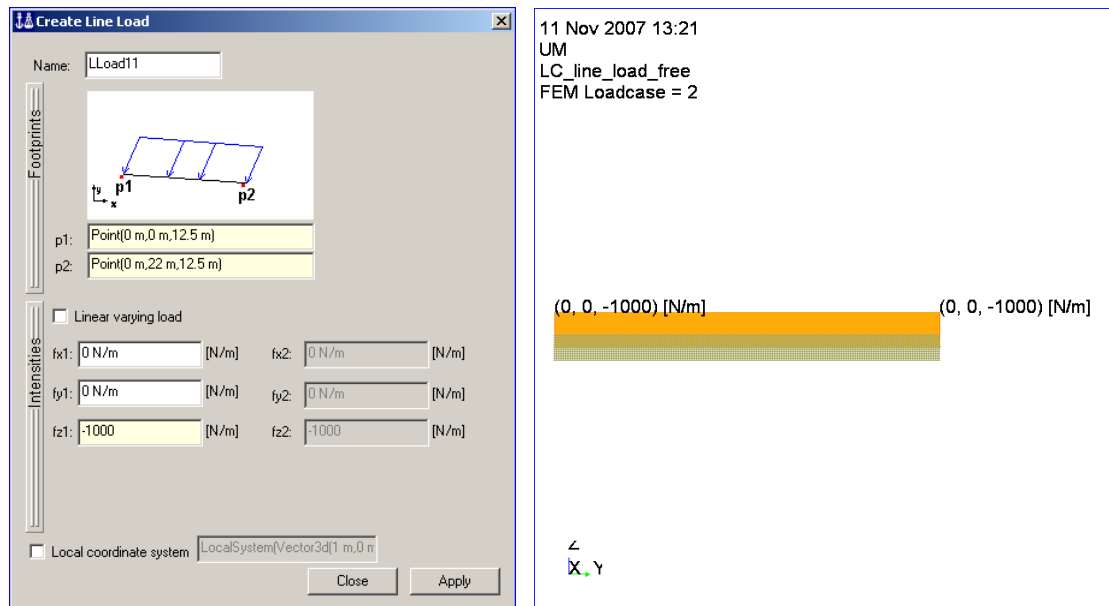
For both options, the line load is defined by using the command **Insert/Explicit Load/Line Load**. Line loads may be inserted by specifying components (F_x , F_y , F_z) in global x,y,z directions or relative to a local x, y, z coordinate system.



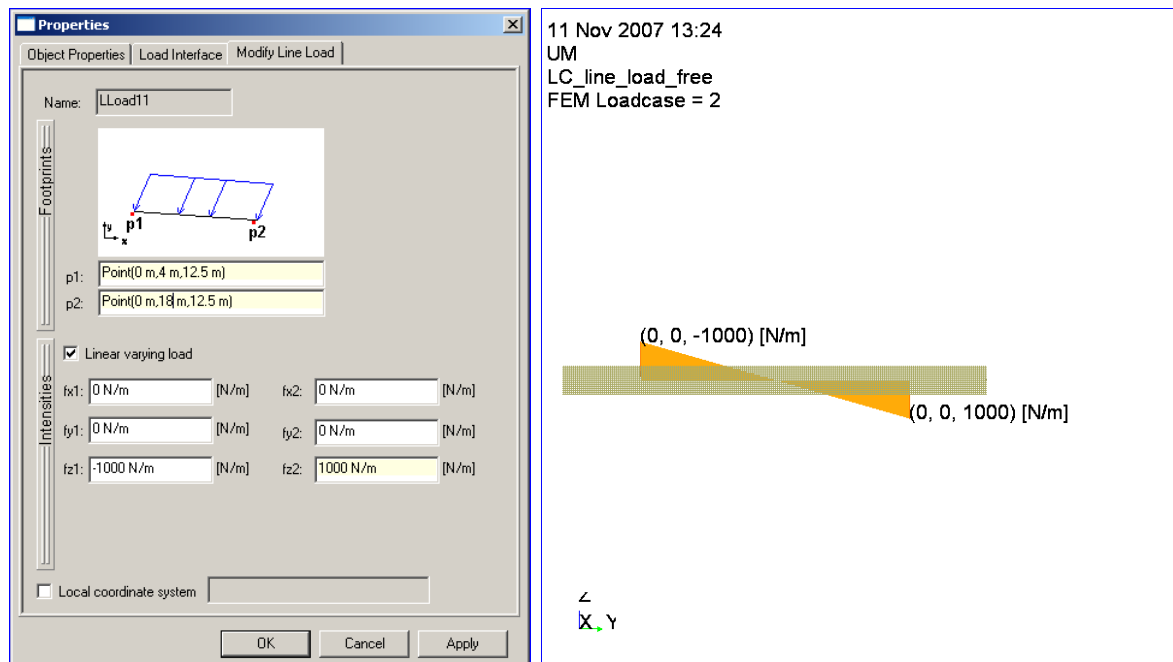
By using features for Javascripting, you can create generic line load functions.

4.2.3.1 The line load as a separate object

This example shows a line load of -1000N in z-direction applied at start and end positions of a beam. Notice that the input dialogue refers to coordinate values that can be typed in manually or found graphic selection.



The line load can be modified by graphic selection, **RMB** and *Properties*. It can also be copied or moved. In this case, the intensity is modified as well as the start and stop positions.



Documenting the line loads is as for point loads, here is shown by selecting *Properties* for the actual load case.

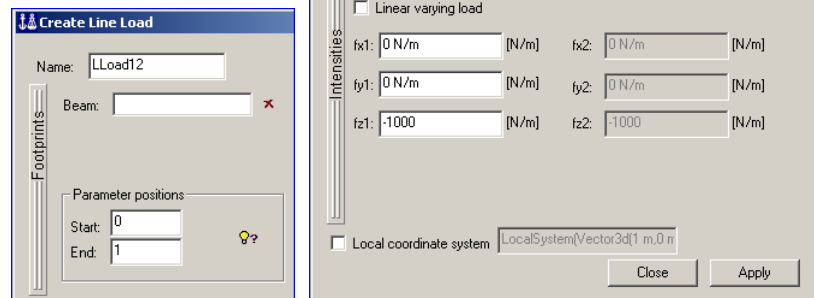
Load Case Properties: LC_line_load_free										
General Equipment Loads Rotation Field Design Condition										
Load	Description	x-coord	y-coord	z-coord	fx	fy	fz	mx	my	mz
LLoad11	Line Line Load.pos1	0	4	12.5	0	0	-1000			
LLoad11	Line Line Load.pos2	0	18	12.5	0	0	1000			

4.2.3.2 The line load referencing the beam or stiffener

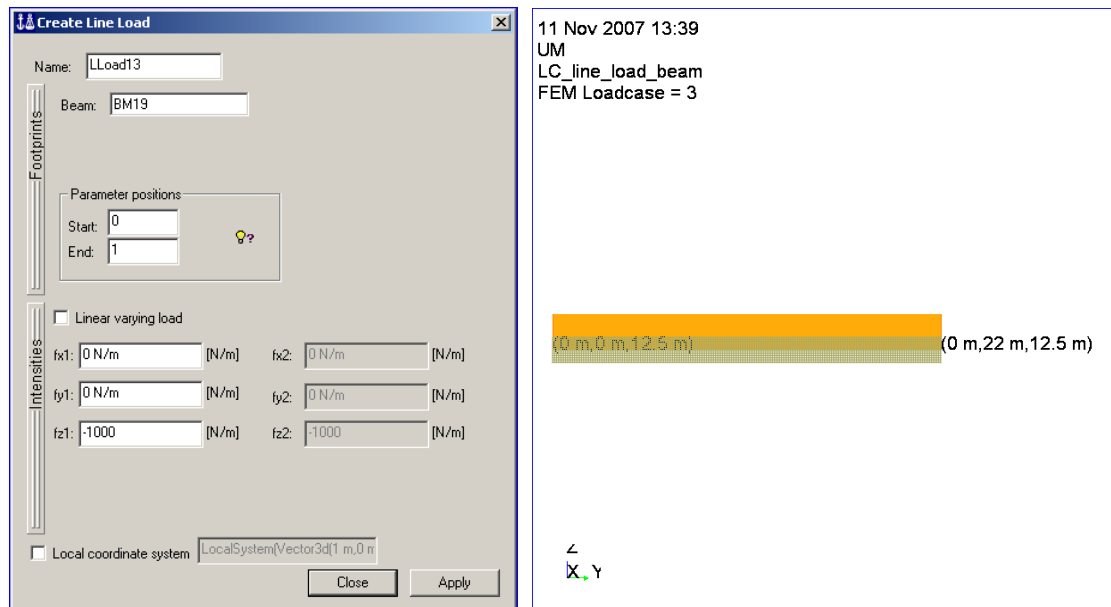
It is also possible to make a line load so that it refers to a beam or stiffener. This means when a beam is moved, the line load is also moved in the same operation. For the previous option, it is necessary to also move the line load similarly to the beam move operation.

To switch to the option where line loads refer to a beam you need to decide the *Footprint* of the line load. In the dialogue box for the line load, drag the vertical bar to the right until you see the input options for the Footprint.

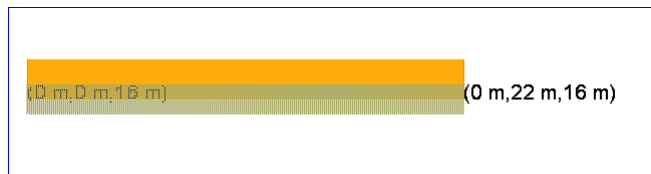
Then change footprint from Line to Beam as shown to the right. The input dialogue box now uses a beam name as reference.



In the example below a typical line load has been inserted on beam Bm19.



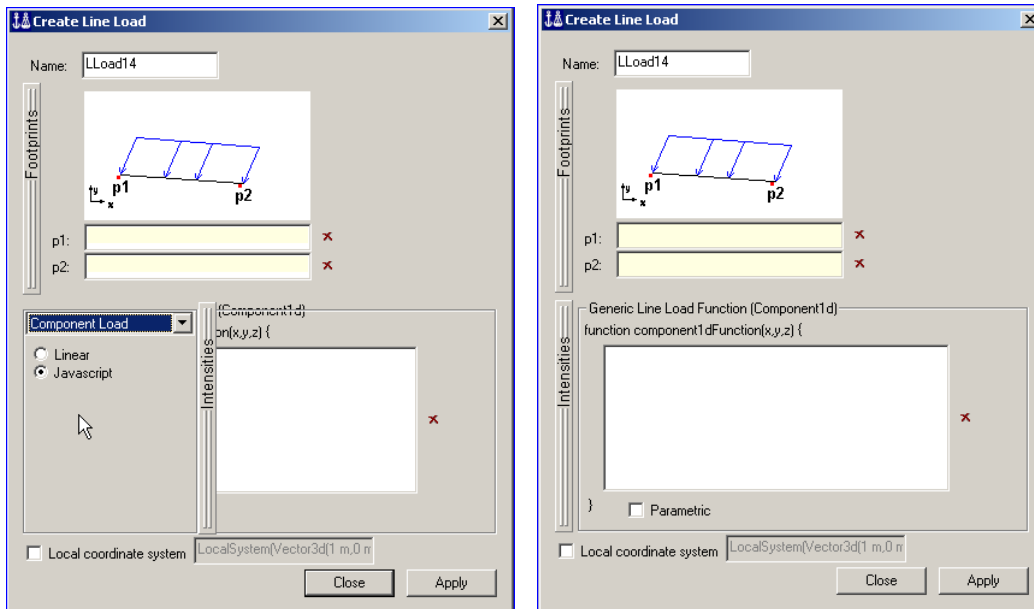
When selecting the beam and move it e.g. 3.5 m in vertical z-direction, the line load follows. Remember to refresh the graphics if you do not see immediate change of line load position.



You can document the line loads as shown on previous pages.

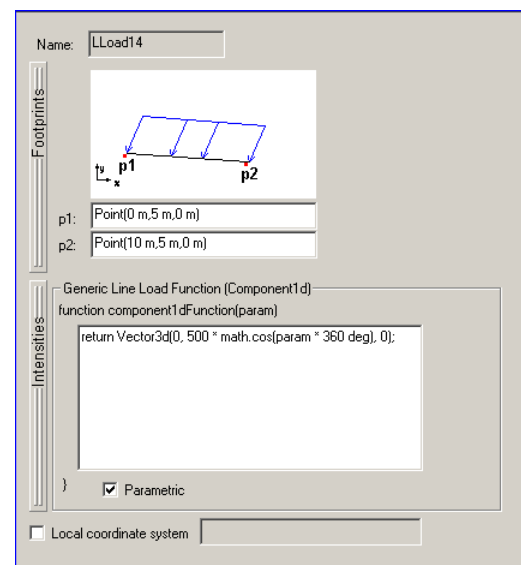
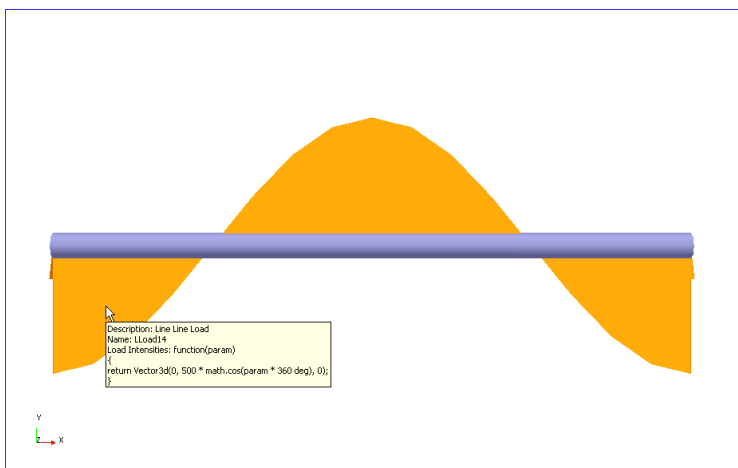
4.2.3.3 Generic line loads

In case you have a line load that is not constant or linearly varying, you can use the feature generic *Line Load Function*. To switch to this feature, drag the vertical *Intensity Bar* until you see the input parameters for *Component Load*. By checking the option for Javascript you have access to advanced line load definition. It is required that you have knowledge on Javascript as you need to make your own function describing the line load.



In the example below two line loads have been applied to the same beam. The first one is a cosine distribution in horizontal direction while the other is a sinus distribution in vertical direction.

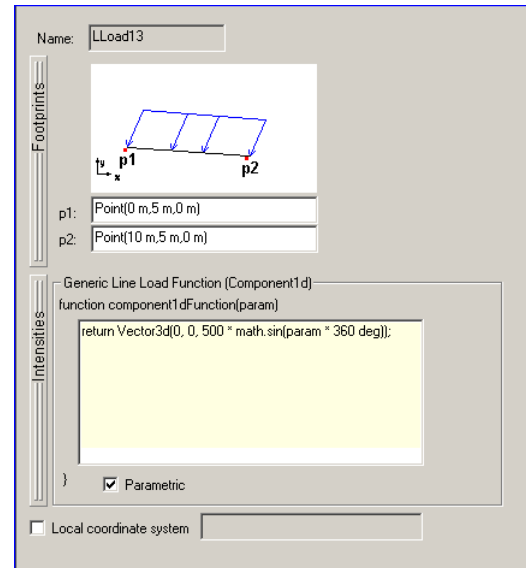
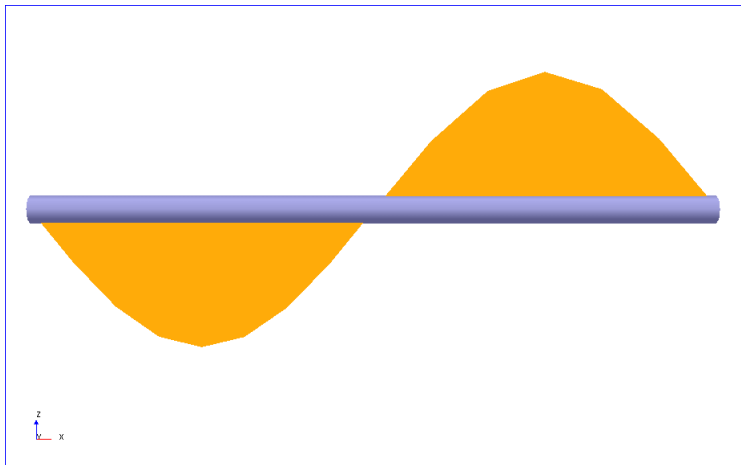
For both options, a parametric representation is used. This means that the length of the beam is parameterized between 0 and 1.



The load is described as $F_y = 500 \cdot \cos(x \cdot 360)$ where $0 \leq x \leq 1$.

The corresponding Javascript command is: `return Vector3d(0, 500 * math.cos(param * 360 deg), 0);`.

Similarly, for a vertical load:



The load is described as $F_z = 500 \cdot \sin(x \cdot 360)$ where $0 \leq x \leq 1$.

The corresponding Javascript command is: `return Vector3d(0,0, 500*math.sin(param*360 deg));`.

For both loads, the angular unit is degrees in this case. You may learn more about mathematical functions from the Help pages under Jscript commands “Math” (you find this under the chapter “Other”).

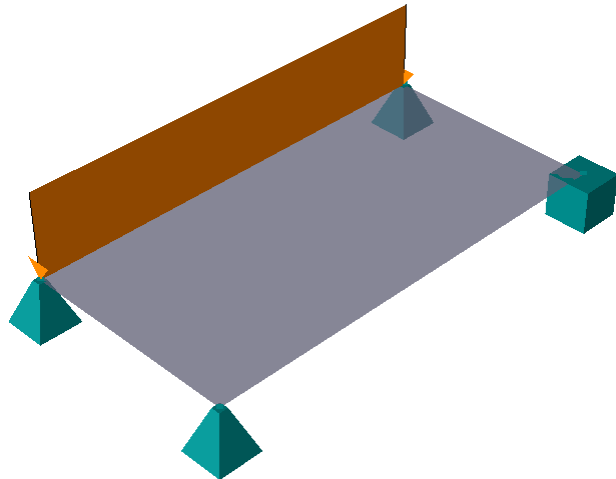
<p>Copyright (c) 1999-2008 DNV Software</p> <p>Introduction Introduction Release Notes Support Request</p> <p>User's Guide Vol 1 - Concept engineering Vol 2 - Waves, pile and soil Vol 3 - Plate/Shell Structures Vol 4 - Beam code checking Reference Documents</p> <p>Command Reference JScript commands</p> <p>Tutorials Example Index</p> <p>Wizards Wizard templates</p> <p>HowTo-Videos Video Index</p>	<h2>Math</h2> <table border="1"> <thead> <tr> <th colspan="2">Function Summary</th> </tr> </thead> <tbody> <tr> <td>double</td> <td>abs(double x) Returns the absolute value of x</td> </tr> <tr> <td>double</td> <td>acos(double x) Returns the arc cosine of x.</td> </tr> <tr> <td>double</td> <td>asin(double x) Returns the arc sine of x</td> </tr> <tr> <td>double</td> <td>atan(double x) Returns the arc tangent of x</td> </tr> <tr> <td>double</td> <td>atan2(double y, double x) Returns the arc tangent of the quotient y/x of the arguments y and x)</td> </tr> <tr> <td>double</td> <td>ceil(double x) Returns the smallest (closest to -∞) number value that is not less than x and is equal to a mathematical integer</td> </tr> <tr> <td>double</td> <td>cos(Quantity x) Returns the cosine of x</td> </tr> <tr> <td>double</td> <td>cos(UnitValue)</td> </tr> <tr> <td>double</td> <td>cos(double)</td> </tr> <tr> <td>double</td> <td>cosh(double x) returns the hyperbolic cosine of x</td> </tr> <tr> <td>double</td> <td>exp(double) Returns the exponential function of x</td> </tr> </tbody> </table>	Function Summary		double	abs (double x) Returns the absolute value of x	double	acos (double x) Returns the arc cosine of x.	double	asin (double x) Returns the arc sine of x	double	atan (double x) Returns the arc tangent of x	double	atan2 (double y, double x) Returns the arc tangent of the quotient y/x of the arguments y and x)	double	ceil (double x) Returns the smallest (closest to -∞) number value that is not less than x and is equal to a mathematical integer	double	cos (Quantity x) Returns the cosine of x	double	cos (UnitValue)	double	cos (double)	double	cosh (double x) returns the hyperbolic cosine of x	double	exp (double) Returns the exponential function of x
Function Summary																									
double	abs (double x) Returns the absolute value of x																								
double	acos (double x) Returns the arc cosine of x.																								
double	asin (double x) Returns the arc sine of x																								
double	atan (double x) Returns the arc tangent of x																								
double	atan2 (double y, double x) Returns the arc tangent of the quotient y/x of the arguments y and x)																								
double	ceil (double x) Returns the smallest (closest to -∞) number value that is not less than x and is equal to a mathematical integer																								
double	cos (Quantity x) Returns the cosine of x																								
double	cos (UnitValue)																								
double	cos (double)																								
double	cosh (double x) returns the hyperbolic cosine of x																								
double	exp (double) Returns the exponential function of x																								

4.2.4 Line loads on plate edges

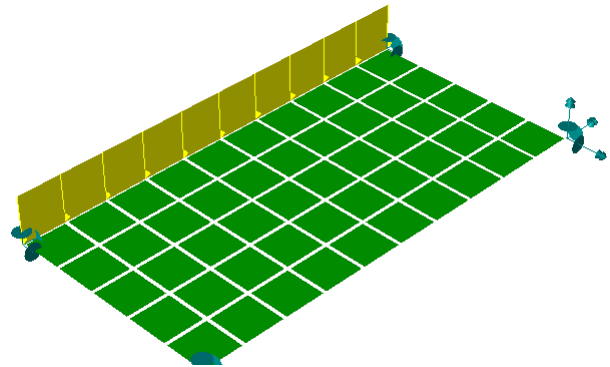
Line loads can be applied along a plate edge.

If you try to apply a line load in the middle of a plate where there is no edge, the line load will not be included in the analysis.

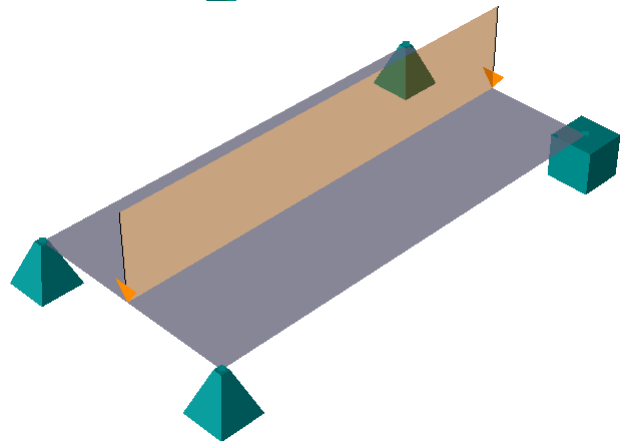
In the illustration to the right a line load is applied to the edge of a plate.



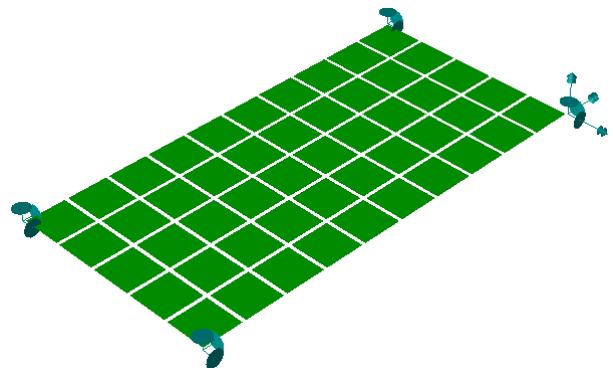
A quick look at “Mesh – All” shows that the load is applied as one would expect.



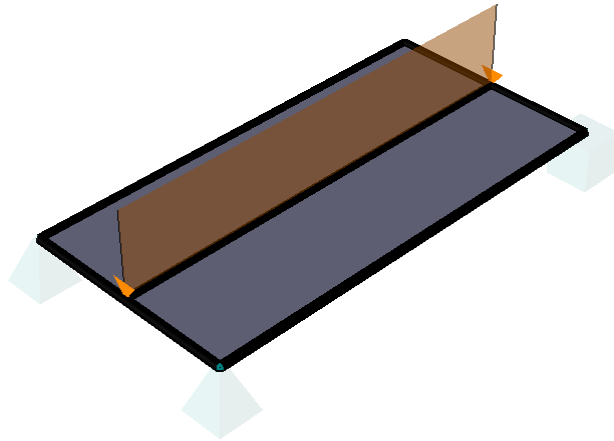
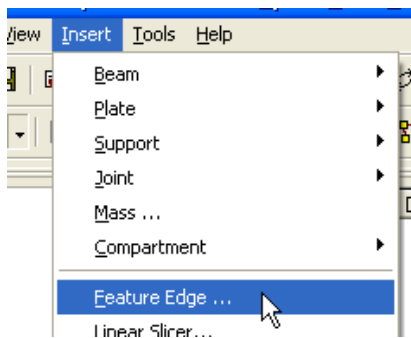
A line load is applied somewhere in the middle of a plate where there are no edges present:



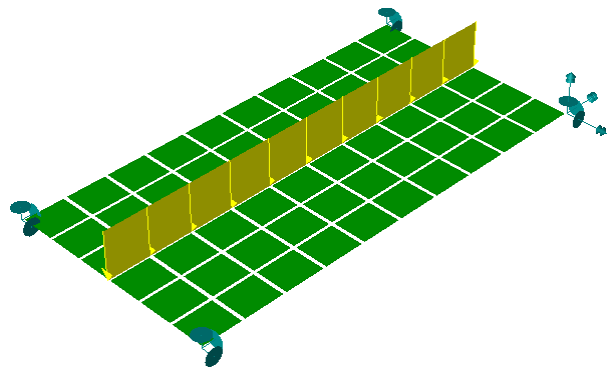
A quick look at “Mesh – All” shows that the load is not applied.



It is sufficient to add a feature edge along the line load to include the line load in the analysis.



After having added a feature edge along the line load and having re run the analysis, the line load shows up on the “Mesh – All” view:

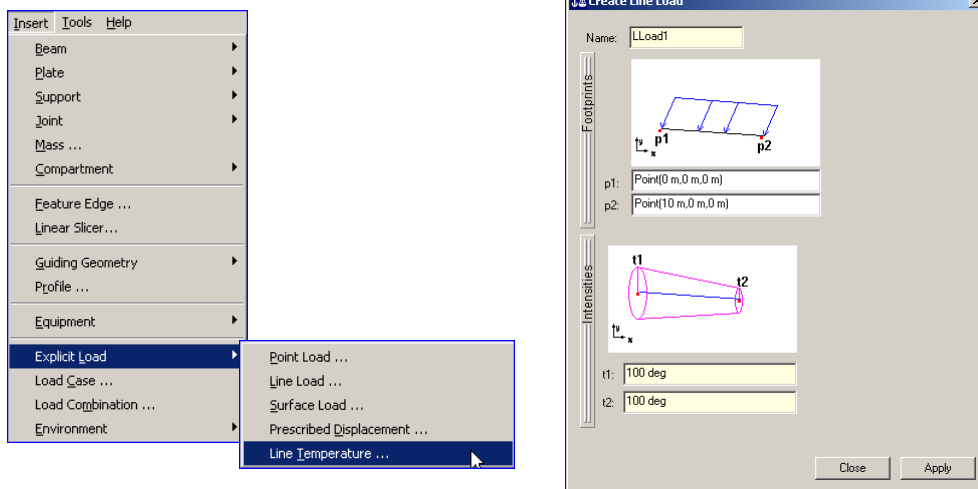


4.2.5 Temperature loads

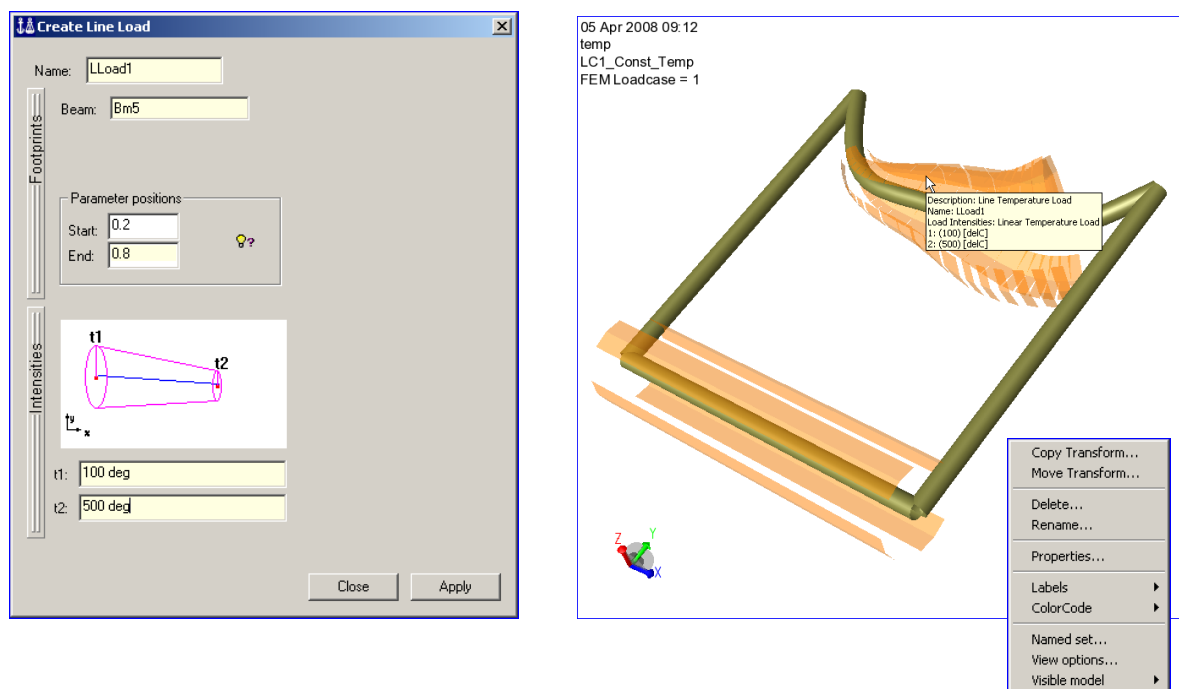
Temperature loads are applied to beams, and as for line loads the temperature loads may be defined as own objects or they may belong to a beam.

The temperature may be constant or linearly varying. You may also use the Javascript option to define other temperature loads. Observe that the temperature load may vary along a beam axis and not across the section. The temperature may also be applied to parts of the beam only.

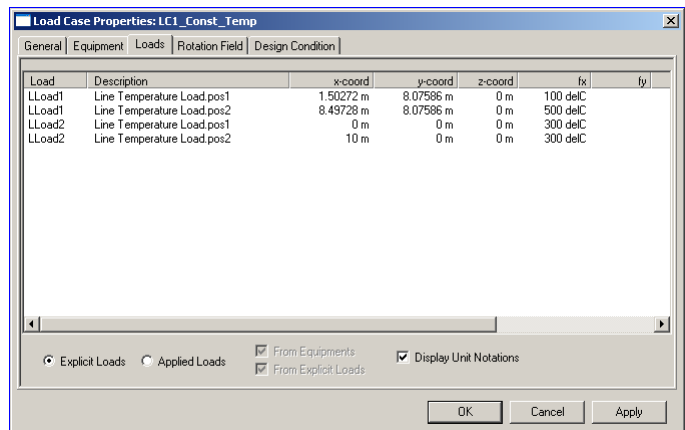
A temperature load is defined from *Insert/Explicit Load/Line Load*.



The example below shows a constant and linearly varying temperature loads applied to two beams. The load may be labelled, modified and deleted from the graphics window (select the load, **RMB** and choose one of the available options).



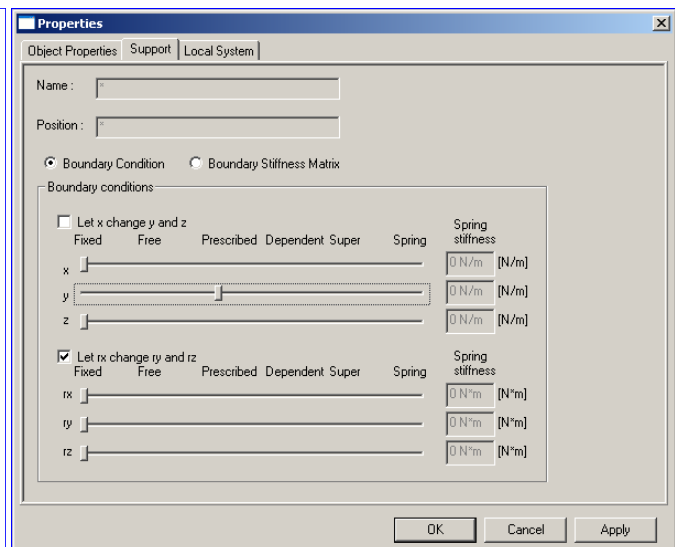
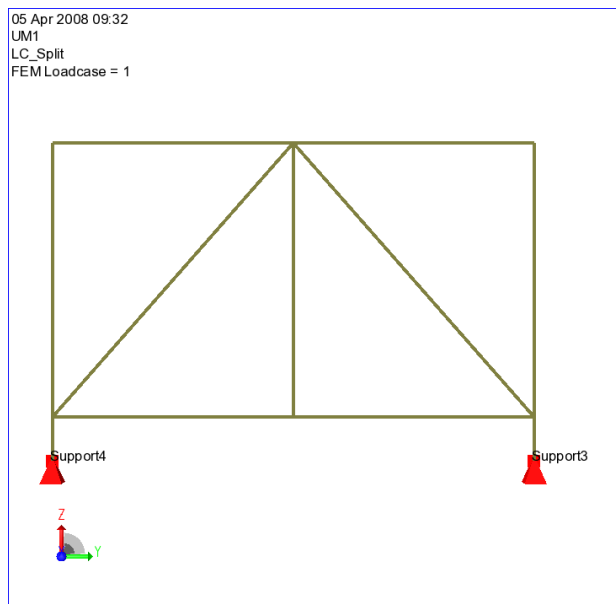
The temperature loads may also be verified from the load case property dialogue box. Select the loadcase, **RMB** and select *Properties*. The temperature load intensities are found under the folder *Loads*. Remember to tick the “Display Unit Notations” if you want to see the units.



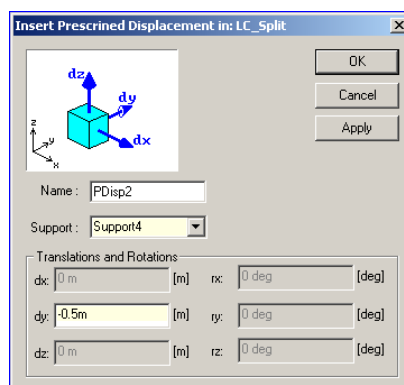
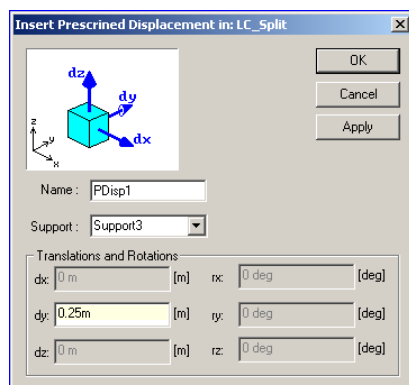
4.2.6 Prescribed displacements

A prescribed displacement is a boundary condition combined with the actual displacement or rotation per loadcase. A support point needs to be inserted first before the actual displacement or rotation can be defined, see the next Chapter on how to define boundary conditions.

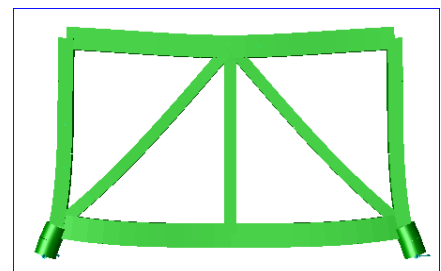
This means that a structure may have several prescribed displacements to the same support point, but in different loadcases. The highlighted supports below are defined as prescribed displacement in x direction



The prescribed displacement per loadcase is applied from **Insert/Explicit Load/Prescribed Displacement**. In this case Support3 receives $\delta_y=0.25\text{m}$ while Support4 is given $\delta_y=-0.5\text{m}$.



Results from analysis shown below.



The saved report may be used to verify the prescribed displacements by investigating the boundary conditions under the Tab *Supports* and the translation (or rotation) under the Tab *Loadcases*, see below.

SupportBoundaryCondition

Name	Description	X [m]	Y [m]	Z [m]	X-Tra	Y-Tra	Z-Tra	X-Rot	Y-Rot	Z-Rot
Support1	Support Point	0	0	0	-2 Fixed	Fixed	Fixed	Free	Free	Free
Support2	Support Point	0	22	0	-2 Fixed	Fixed	Fixed	Free	Free	Free
Support3	Support Point	28	22	0	-2 Fixed	Prescribed	Fixed	Fixed	Fixed	Fixed
Support4	Support Point	28	0	0	-2 Fixed	Prescribed	Fixed	Fixed	Fixed	Fixed

SupportDisplacement

LoadName	Description	LoadType	X [m]	Y [m]	Z [m]	DX [m]	DY [m]	DZ [m]	RX [deg]	RY [deg]
PDisp1	Prescribed Disp	Explicit	28	22	0	-2	0	0.25	0	0
PDisp2	Prescribed Disp	Explicit	28	0	0	-2	0	-0.5	0	0

4.3 Surface loads

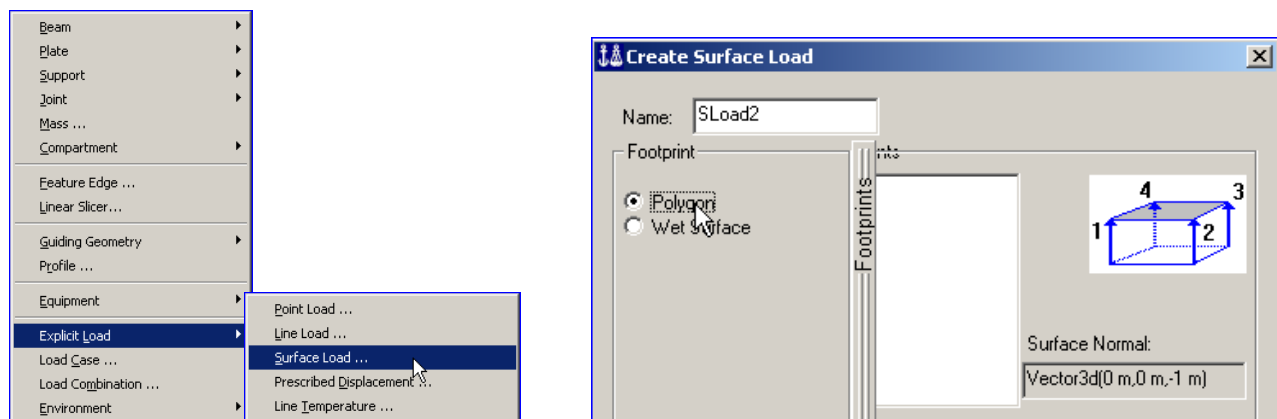
Surface loads are applied to wet surfaces for plates and shells. A wet surface is associated with the plate or shell – this means that when such objects are moved, the load will follow.

For plates it is also possible to specify the surface loads with a polygon footprint. Applied loads are computed when there is an intersection between the footprint and the structure. In this case the loads will not follow when the structural object is moved.

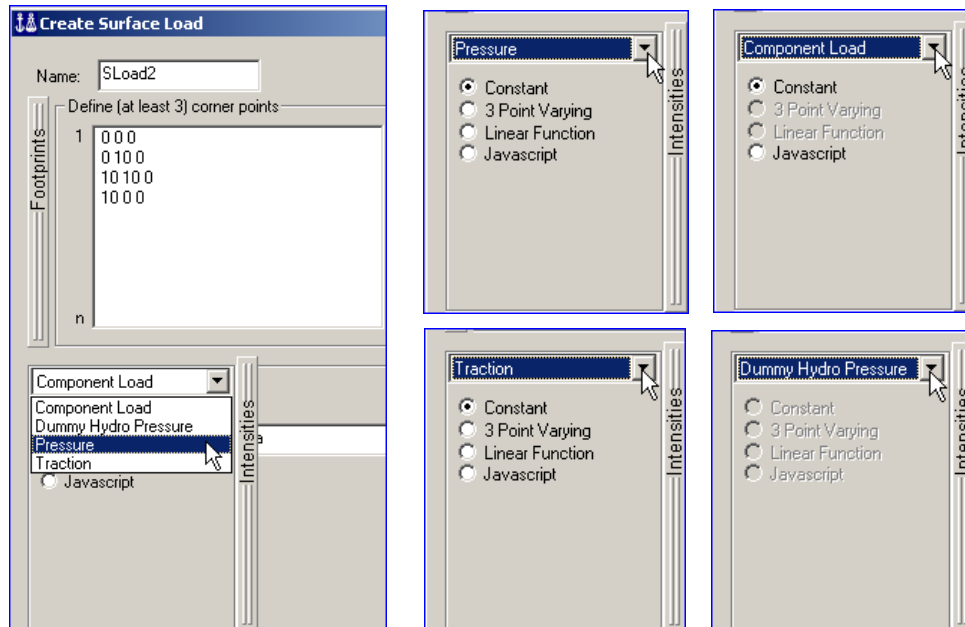
4.3.1 Plate surface loads

These are surface loads defined by using the polygon footprint option. It is a flexible solution for plates as you can make the surface loads independent of the plates or wet surfaces. This option can be used on flat plates. The surface loads may be constant, linear varying, three point varying or defined by a java script command. All options are explained in the following except for the javascript option – see previous chapter on beam line loads on how to do this.

A surface load is defined from the pulldown menu **Insert/Explicit Load/Surface Load**. The plate surface loads are normally defined using the polygon footprint – move the slider *Footprints* below to select polygon footprint. Surface loads defined using a wet surface as footprints are documented in the following chapter “Pressure loads on shells”.



A surface load may be either be a pressure (normal to the plate), a traction load (parallel to the plate) or a component load (build up of components in global x, y and z directions). In addition there is a special type “*Dummy Hydro Pressure*” being used to define the extent of a hull and internal tanks subjected to hydrodynamic loads and pressure. Normally this load type is used when the footprint is the wet surface option, see next chapter.

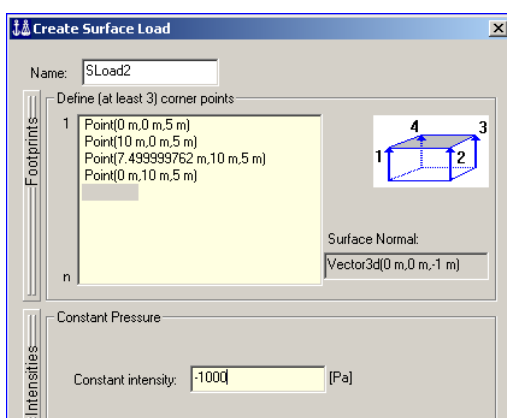


Each of the options is described in the following except for “Dummy Hydro Pressure” which is documented in the chapter “Transfer pressure data to HydroD”.

4.3.1.1 Pressure loads

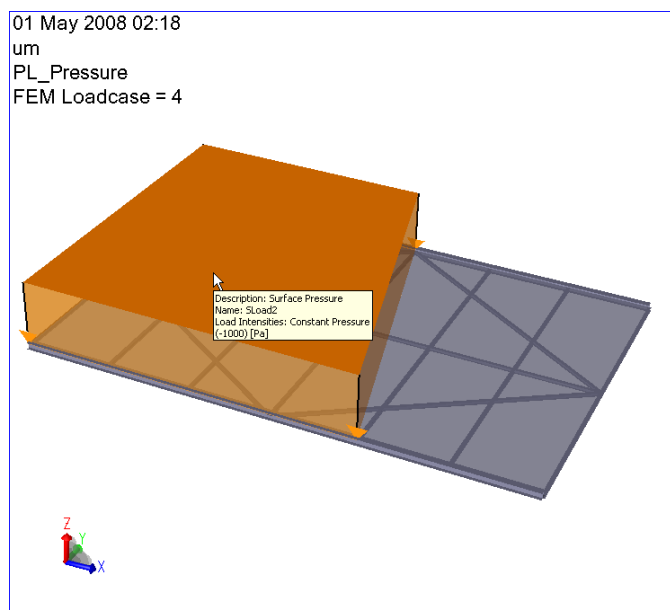
Select surface load to type *Pressure*. You may choose between Constant, 3 Point Varying, linear function and Javascript. The three first options are shown in the following. The pressure is normal to the footprint that it is acting on.

Constant pressure load.

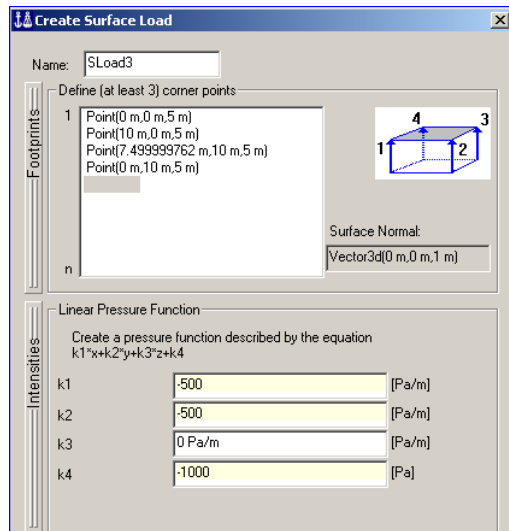


This example shows that the load footprint is independent of plate edges.

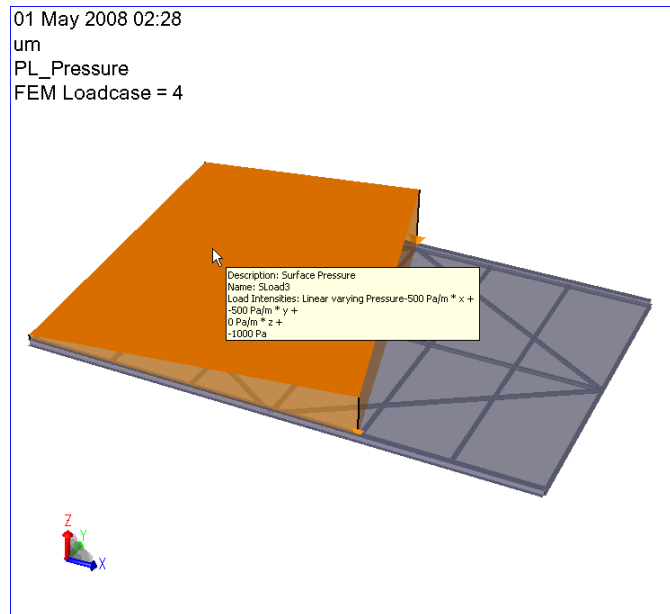
The direction of the pressure load is relative to the footprint surface normal. The right hand rule applies.



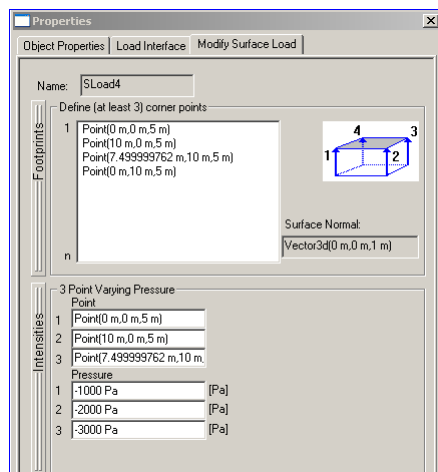
Linear varying pressure.



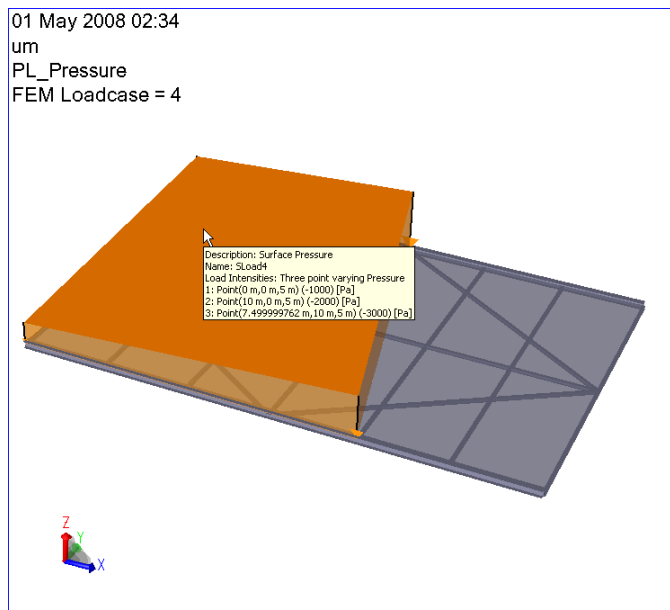
The linear varying load includes a constant part and can vary as a function of global x, y and z coordinates.



Three point varying pressure.



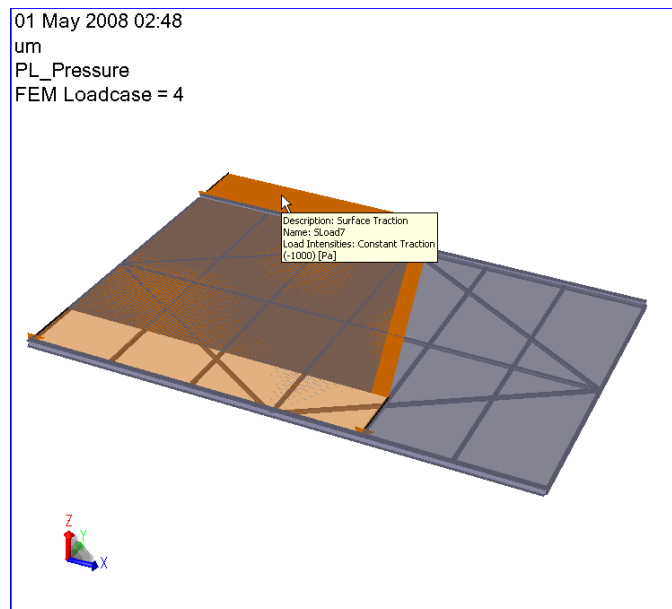
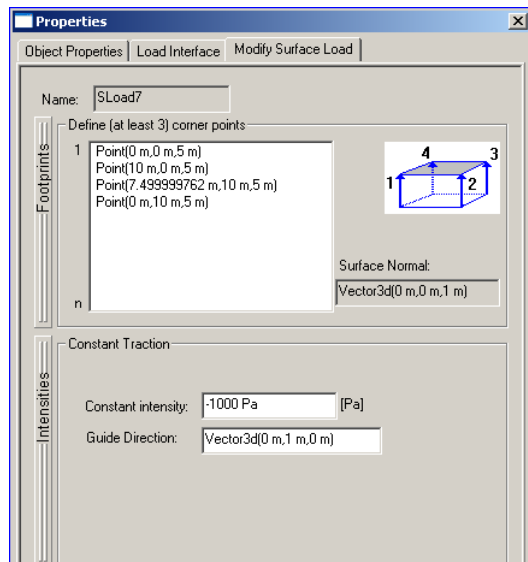
A three point varying load is determined from the load intensities at three defined points.



4.3.1.2 Traction loads

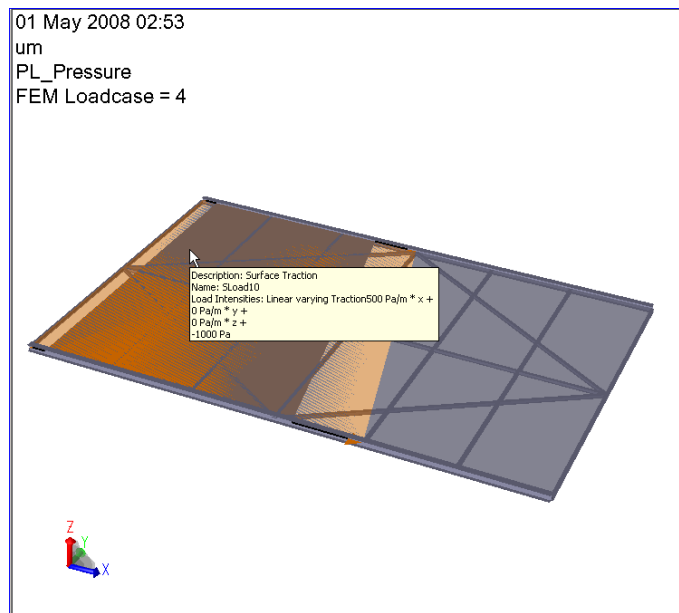
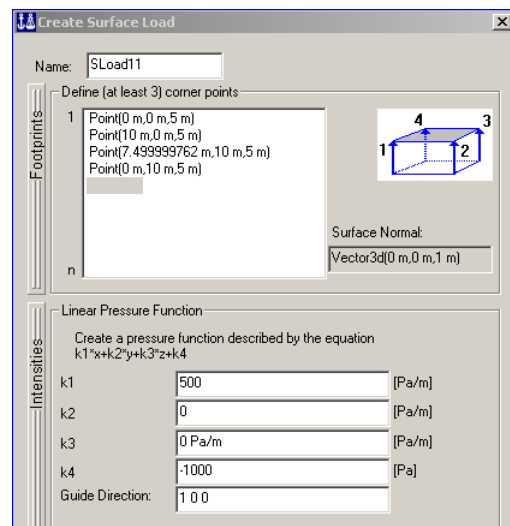
Select surface load to type *Traction*. As for *Pressure*, a *Traction* load can be of type Constant, 3 Point Varying, linear function and Javascript. The three first options are shown in the following. The traction is parallel to the plate that it is acting on.

Constant traction.



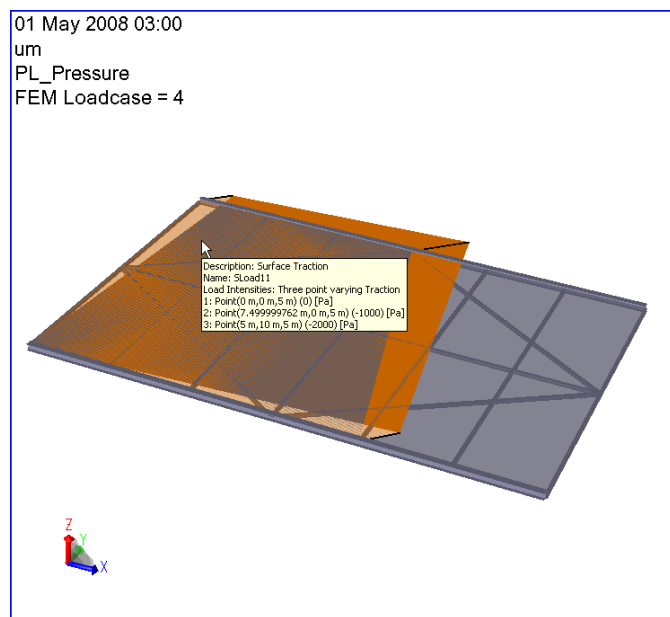
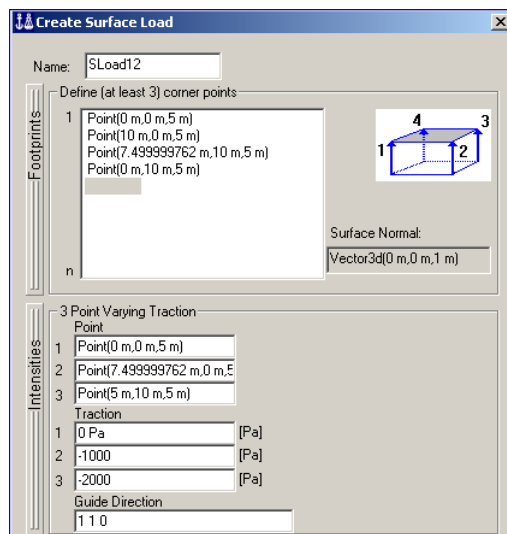
The constant traction load is defined by intensity and a direction. In this case the guide direction is in global y-direction.

Linear varying traction.



As for the constant traction, the linear varying traction load is governed by a direction and intensity consisting of a constant part and variable parts along global x, y or z directions. In this case the guide direction is along the global x-direction.

Three point varying traction.

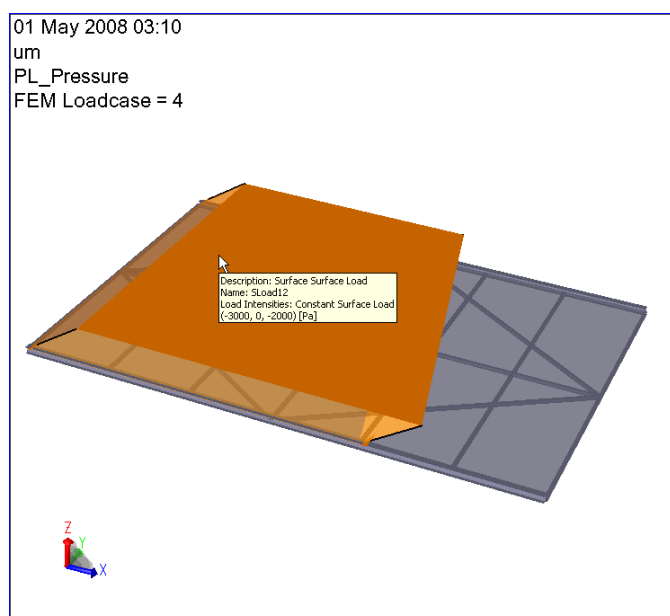
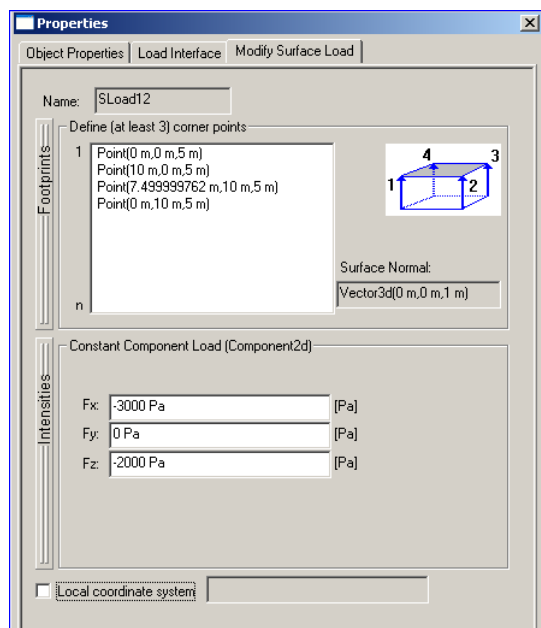


In this case the traction load is defined from load intensities at three points as well as a direction – in this case along 45 degrees (in the middle between global x- and y-directions).

4.3.1.3 Component load

Select surface load to *Component load*. In this case you define loads of type Constant (see example below) or Javascript. A component load is a pressure load built up from load intensities in global x, y, and z-directions or a local coordinate system.

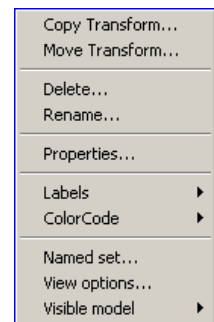
Constant component load.



The component load in this case has constant pressure intensity in both global x and z directions.

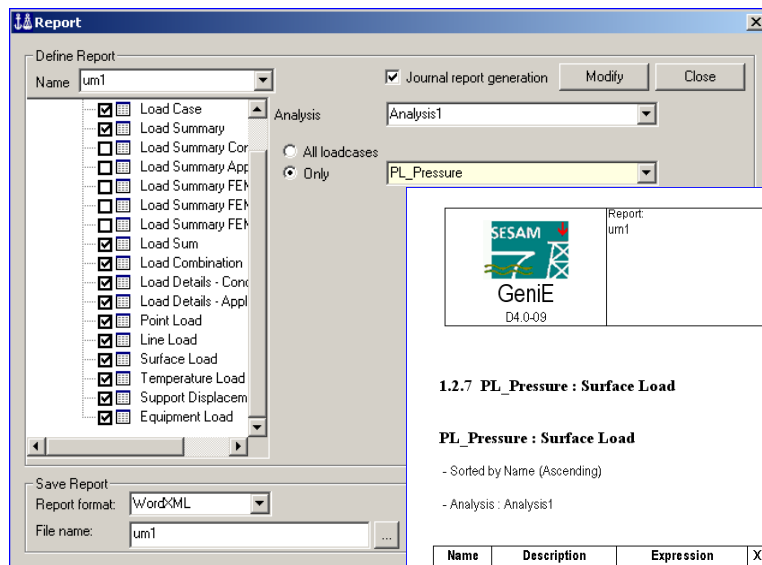
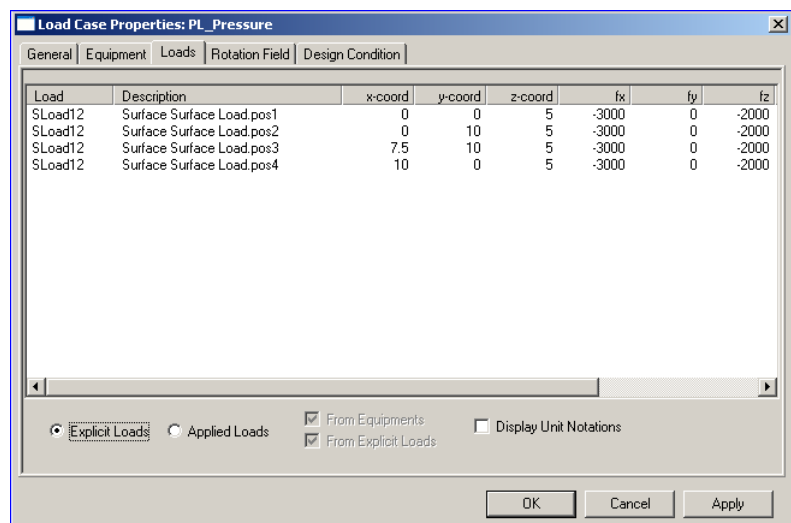
4.3.1.4 Modify and verify surface loads


All of the above loads can be edited by graphic selection of the load, **RMB** and choose *Properties*. You may also copy, move, delete or rename the load.



There are several ways of verifying a load.

- The mouse tool-tip feedback in the graphics window
- From the loads property dialogue box
- From the load case property dialogue box (see the example to the right)
- From a printed report. Use **File/Save Report** and make a report that includes the relevant parts needed to document the loads. See below for an example.



	Report: um1	Model Id: um1	Sign: nek
		Description: um	Date: 01-May-2008
		Model file name: C:\Program Files\DNV\SGeniE_D4009\Workspaces\um	Last saved: 01-May-2008 03:23:24

1.2.7 PL_Pressure : Surface Load

PL_Pressure : Surface Load

- Sorted by Name (Ascending)

- Analysis : Analysis1

Name	Description	Expression	X [m]	Y [m]	Z [m]	Pressure [Pa]	FX [Pa]	FY [Pa]	FZ [Pa]
SLoad12	Surface Surface Load	Constant Surface Load (-3000, 0, -2000) [Pa]	0	0	5		-3000	0	-2000
			10	0	5		-3000	0	-2000
			7.5	10	5		-3000	0	-2000
			0	10	5		-3000	0	-2000

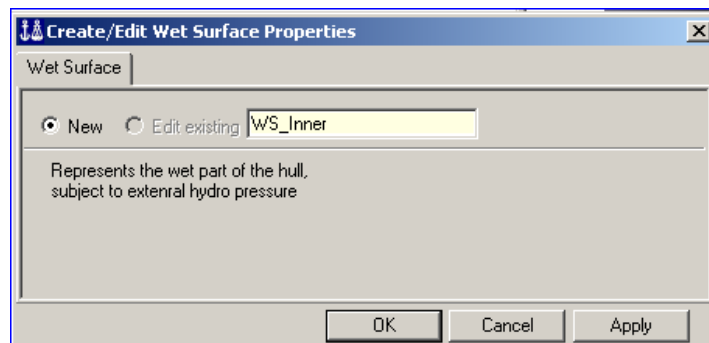
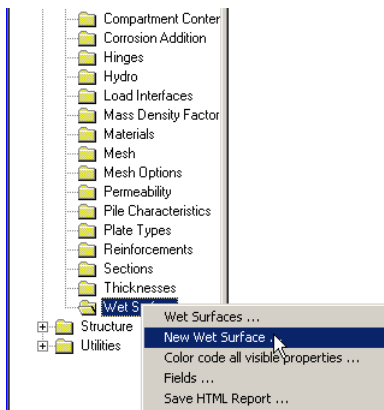
4.3.2 Surface loads on shells

Surface loads on shells can be applied to a wet surface. The wet surface is then used to define the footprint of the loads. In addition to wet surfaces, loads on curved surfaces may be defined if they cover the whole surface of a given named plate.

A tube having two wet surfaces is used to describe how surface loads can be applied to shell structures.

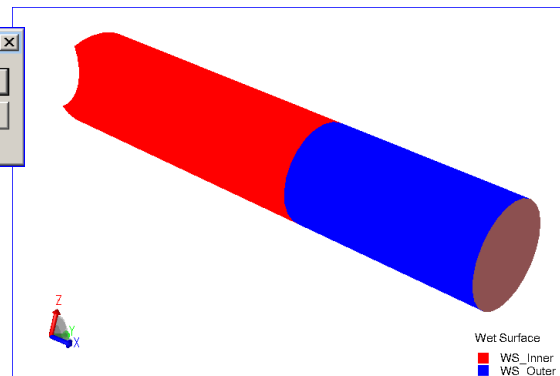
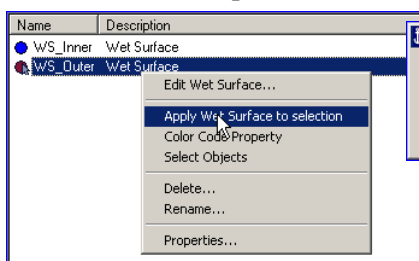
4.3.3 Define wet surfaces

The procedure to make a wet surface is to define it as a property and connect it to a plate/shell surface(s). The property is defined from **Edit/Properties/Wet Surfaces** or from the browser as shown below.

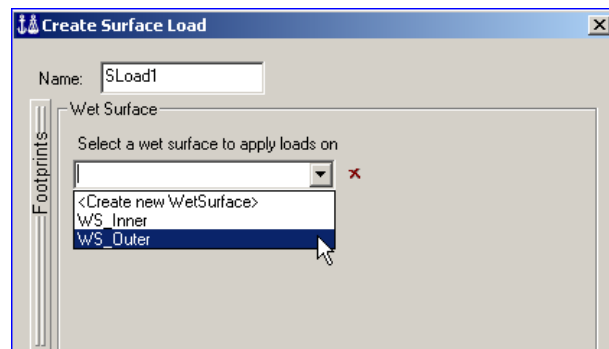
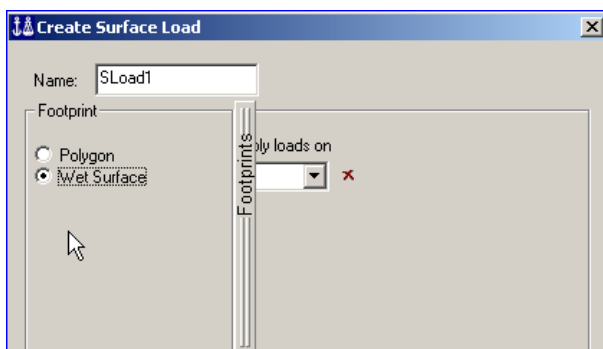


Two wet surfaces are defined; *WS_Inner* and *WS_Outer*.

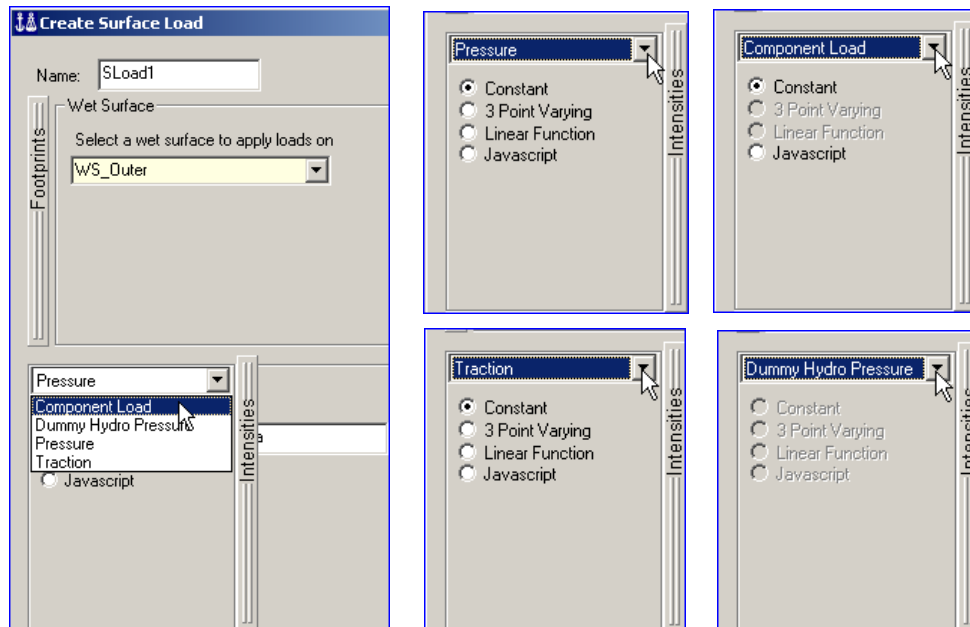
When the wet surfaces are applied to a selection of plate(s)/shell(s) it is necessary to specify which side of the plate to assign the property to. This now determines the direction of the surface loads (or what is inside or outside in a compartment).



The wet surfaces *WS_Inner* and *WS_Outer* will be used as footprints when defining surface loads acting on the shells.



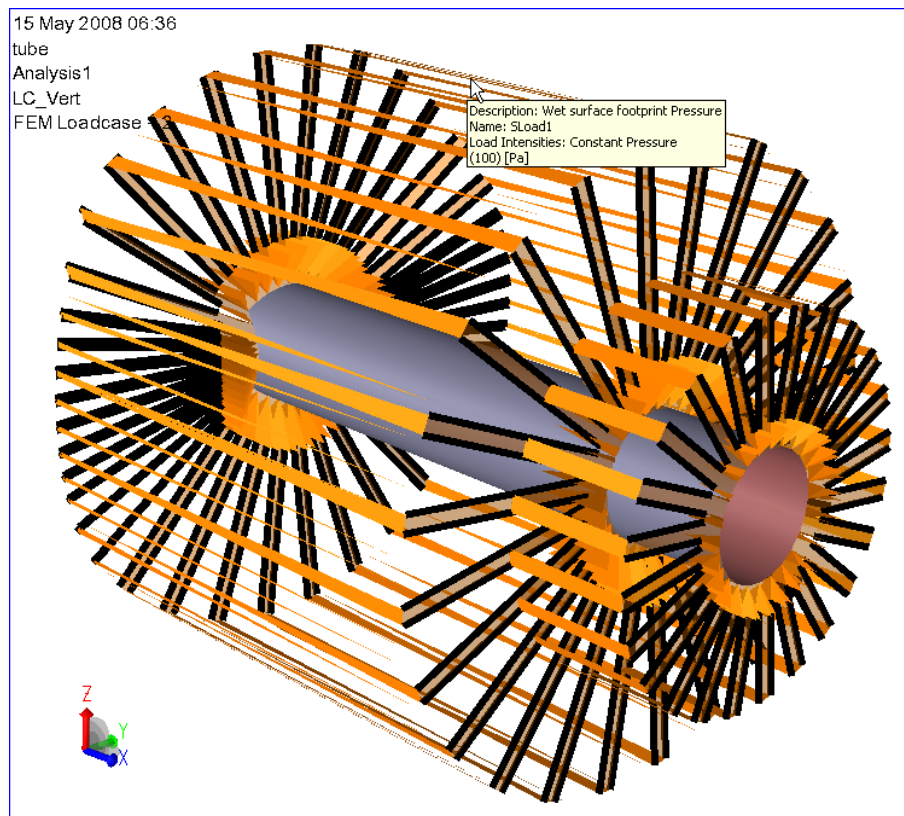
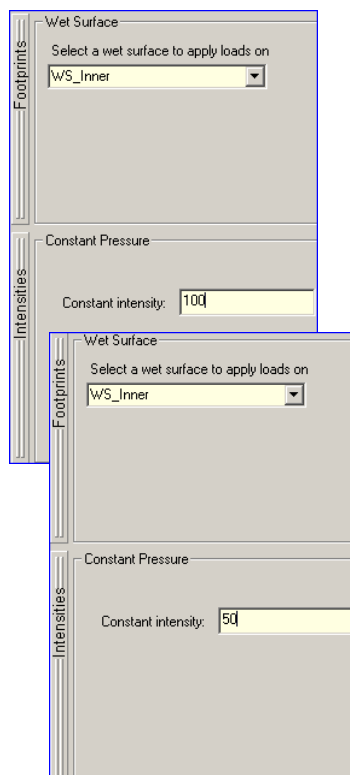
A surface load may be either be a pressure (normal to the wet surface), a traction load (parallel to the wet surface) or a component load (build up of components in global x, y and z directions). In addition there is a special type “*Dummy Hydro Pressure*” being used to define the extent of a hull and internal tanks subjected to hydrodynamic loads and pressure.



4.3.3.1 Pressure loads

Select surface load to type *Pressure*. You may choose between Constant, 3 Point Varying, linear function and Javascript. The three first options are shown in the following. The pressure is normal to the wet surface that it is acting on.

Constant pressure load



3 Point Varying load

Name: SLoad4

Wet Surface

Select a wet surface to apply loads on

WS_Outer

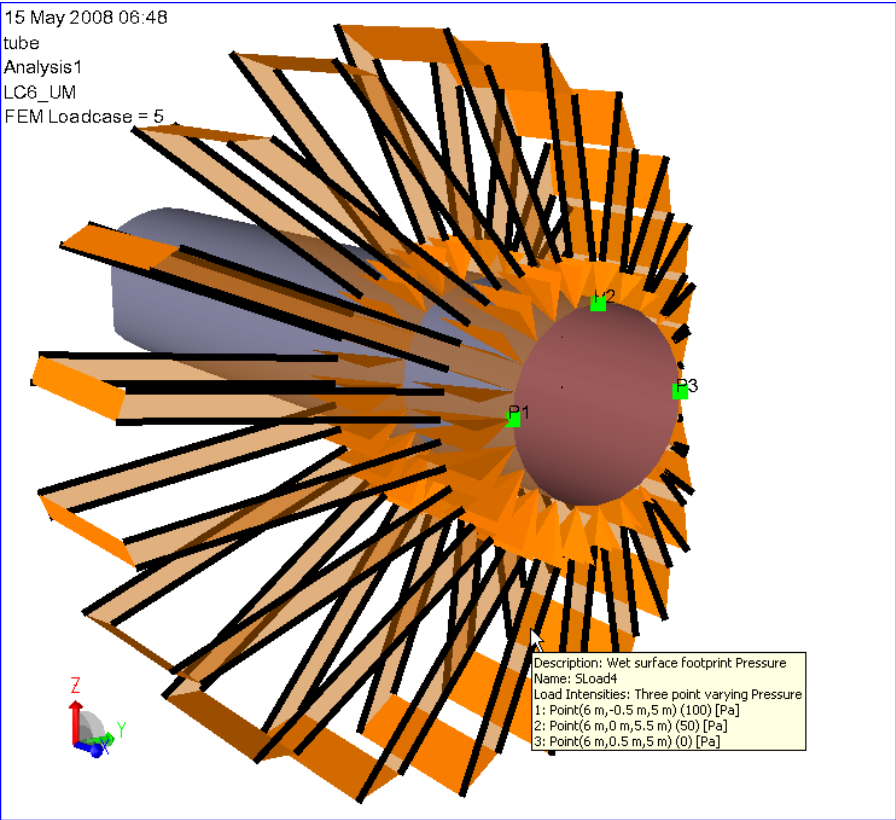
3 Point Varying Pressure

Point

1	Point(6 m,-0.5 m,5 m)
2	Point(6 m,0 m,5.5 m)
3	Point(6 m,0.5 m,5 m)

Pressure

1	100 Pa	[Pa]
2	50 Pa	[Pa]
3	0 Pa	[Pa]



Linear function load

Name: SLoad6

Wet Surface

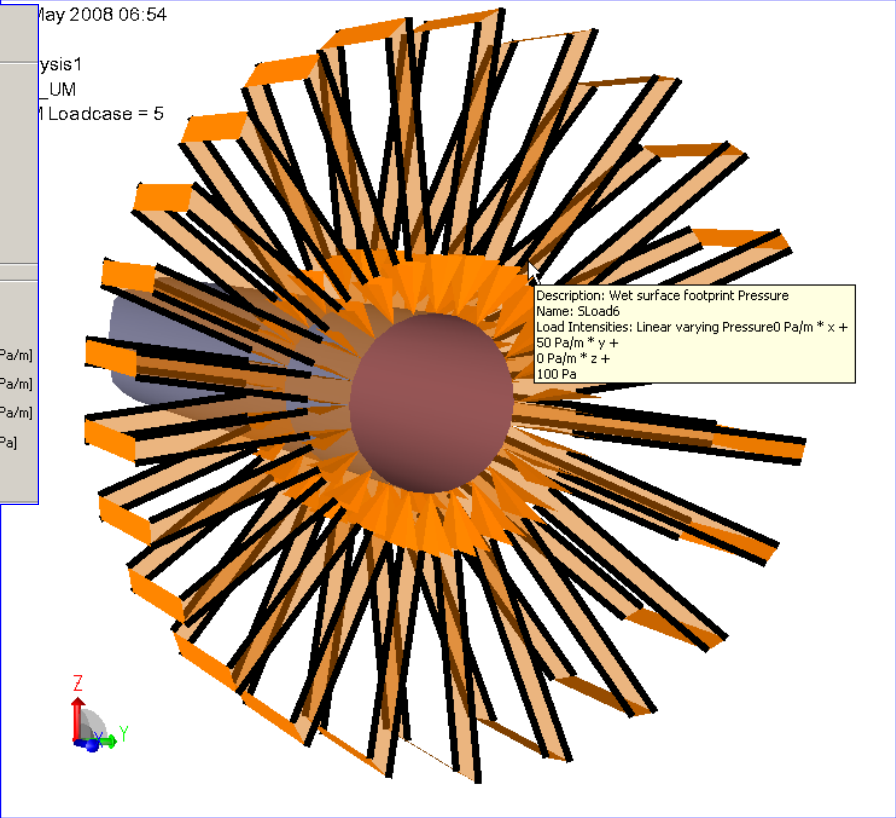
Select a wet surface to apply loads on

WS_Outer

Linear Pressure Function

Create a pressure function described by the equation $k1*x+k2*y+k3*z+k4$

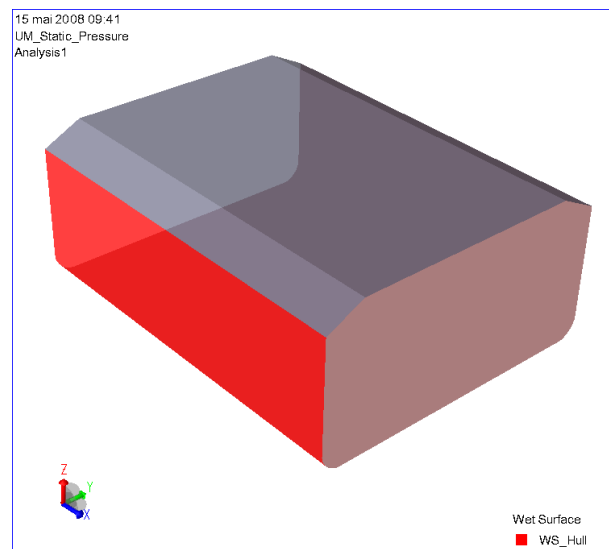
k1	0 Pa/m	[Pa/m]
k2	50 Pa/m	[Pa/m]
k3	0 Pa/m	[Pa/m]
k4	100 Pa	[Pa]



Java script load

This option allows you to make your own function to describe the surface load. A typical example may be static water pressure simulating for example an intact or damaged condition if you do not want to run a hydrostatic or hydrodynamic analysis where such loads are automatically generated. In such cases the wetted surface to be loaded differs from case to case. The below example shows how to create different load conditions based on one wet surface. Please note that such computations may take time as it is necessary to evaluate the java script function at multiple positions. It is required that you have knowledge on Javascript as you need to make your own function describing the surface load.

The model and the wet surface are shown to the right. There are no emphases on defining the boundaries of the wet surface to correspond to the actual water surface – the wet surface has been defines simply as a footprint to use when making the load.



The load to be used in this example is as follows:

Function for generating a hydrostatic pressure on a wet surface for heeling about global x-axis and draught at origin

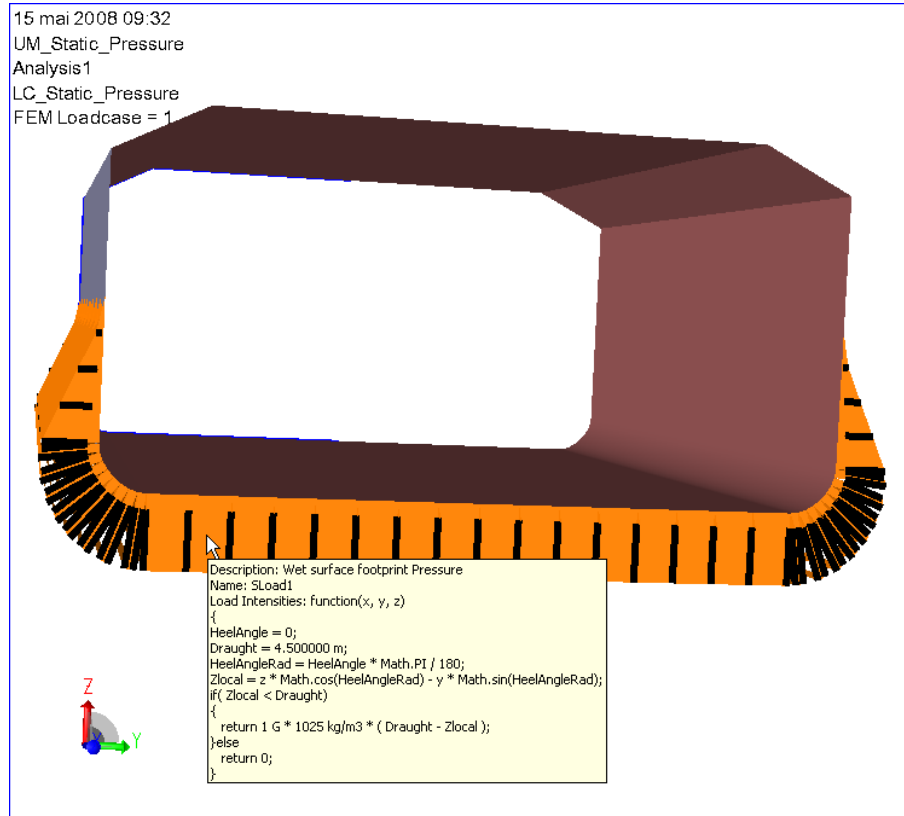
- Make a model (like above)
- Define a wet surface (like above)
- Make a loadcase
- Insert explicit load, surface load, select the wet surface, select pressure and use option Javascript.
- Edit the heel angle (around global x-axis) and draught in the js-file below
- Cut and paste the content of the js-file into the javascript window

```

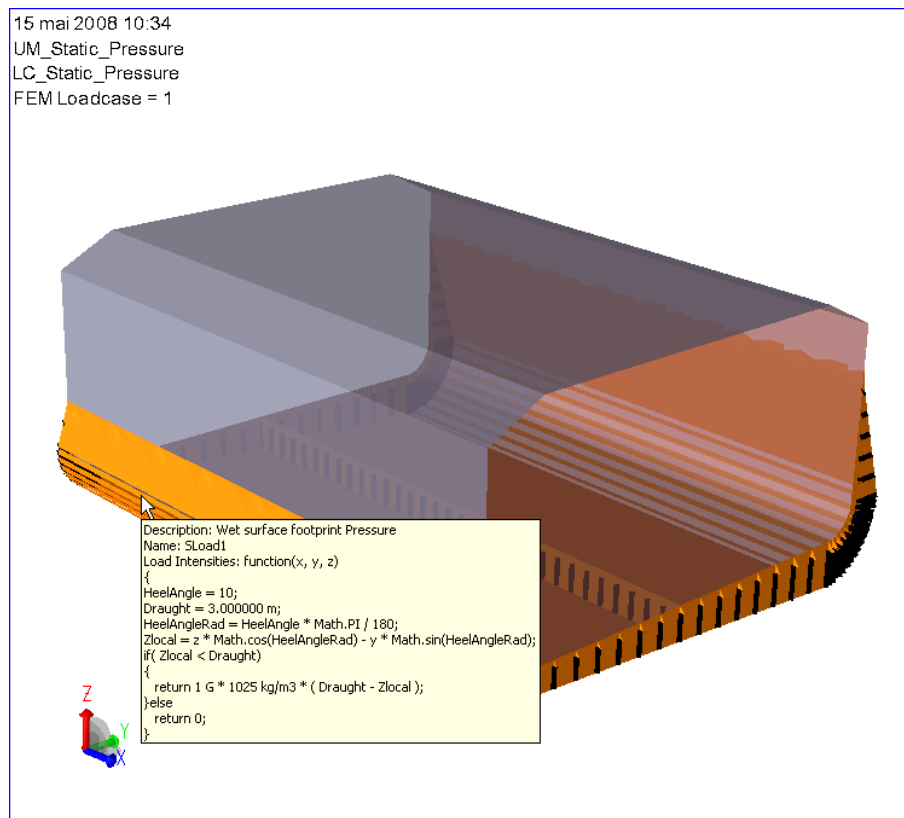
HeelAngle = 20 ;
Draught = 4.5 m;
//
HeelAngleRad =(HeelAngle*Math.PI)/180;
Zlocal=(z*Math.cos(HeelAngleRad))-(y*Math.sin(HeelAngleRad));
if ( Zlocal < Draught)
{
return 1 G * 1025 kg/m3 * (Draught - Zlocal);
}
else return 0;

```

The picture to the right shows a condition with *Draught* = 4.5 m and *HeelAngle* = 0 deg.



In this case, the parameters have been modified to *Draught* = 3.0 m and *HeelAngle* = 10 deg.



4.3.3.2 Traction loads

Select surface load to type *Traction*. You may choose between Constant, 3 Point Varying, linear function and Javascript. The first option is shown below; the three remaining options are similar to this shown in the Section *Pressure loads*. The traction is parallel to the wet surface that it is acting on.

Constant traction load

Footprints

Wet Surface

Select a wet surface to apply loads on

WS_Outer

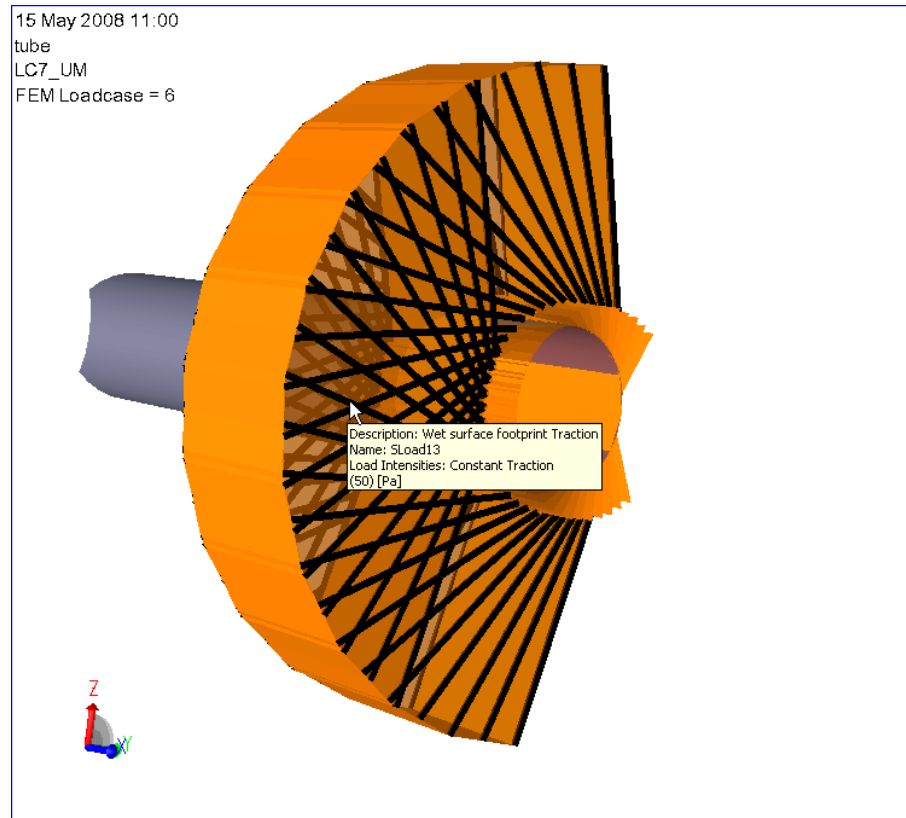
Intensities

Constant Traction

Constant intensity: 50 [Pa]

Guide Direction: 0 1 0

The direction of the traction load is determined by the Guide Direction; in this case the global Y-direction.



Footprints

Wet Surface

Select a wet surface to apply loads on

WS_Outer

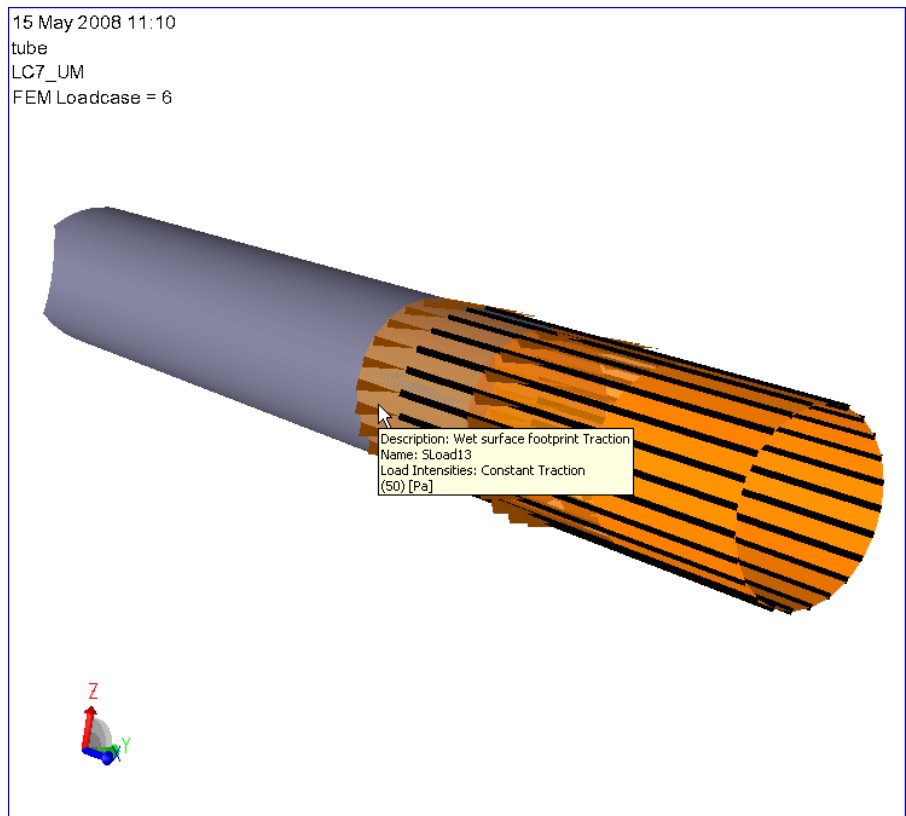
Intensities

Constant Traction

Constant intensity: 50 Pa [Pa]

Guide Direction: Vector3d(-1 m,0 m,0 m)

The direction of the traction load is determined by the Guide Direction; in this case the global X-direction (in negative direction).



4.3.3.3 Component loads

Select surface load to type *Component Load*. You may choose between Constant and Javascript. The first option is shown below; the Javascript option is similar to this shown in the Section *Pressure loads*. A component load will set up a constant pressure build up by components in global x, y and z-directions.

Constant component load

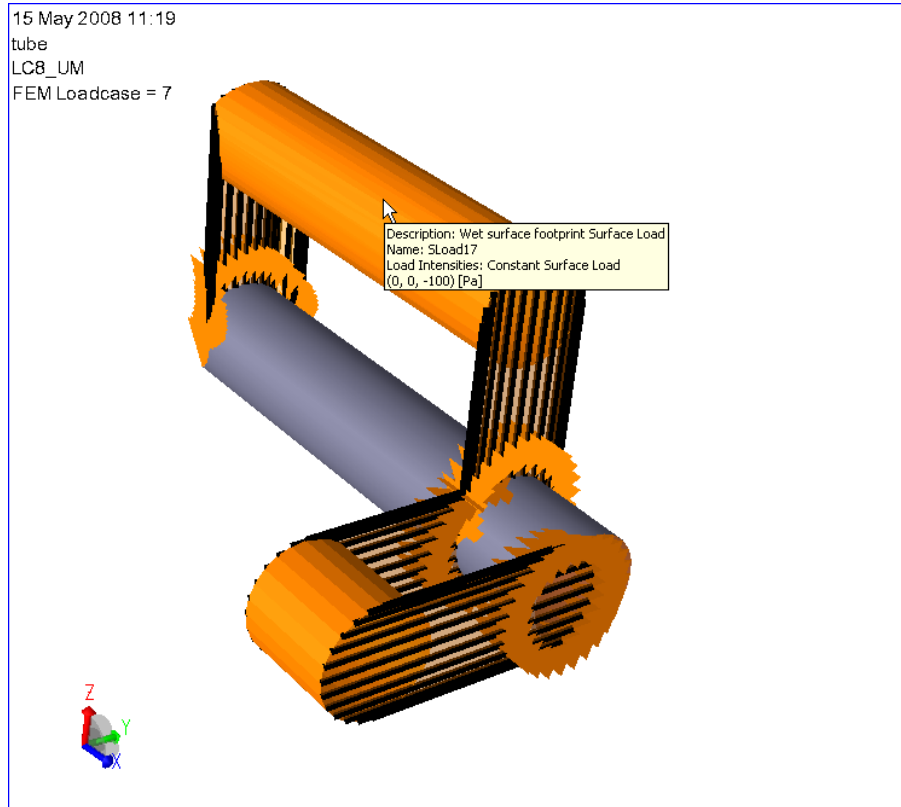
15 May 2008 11:19
tube
LC8_UM
FEM Loadcase = 7

Footprints
Wet Surface
Select a wet surface to apply loads on
WS_Inner

Intensities
Constant Component Load (Component2d)
Fx: 0 Pa [Pa]
Fy: 0 Pa [Pa]
Fz: -100 Pa [Pa]

Footprints
Wet Surface
Select a wet surface to apply loads on
WS_Outer

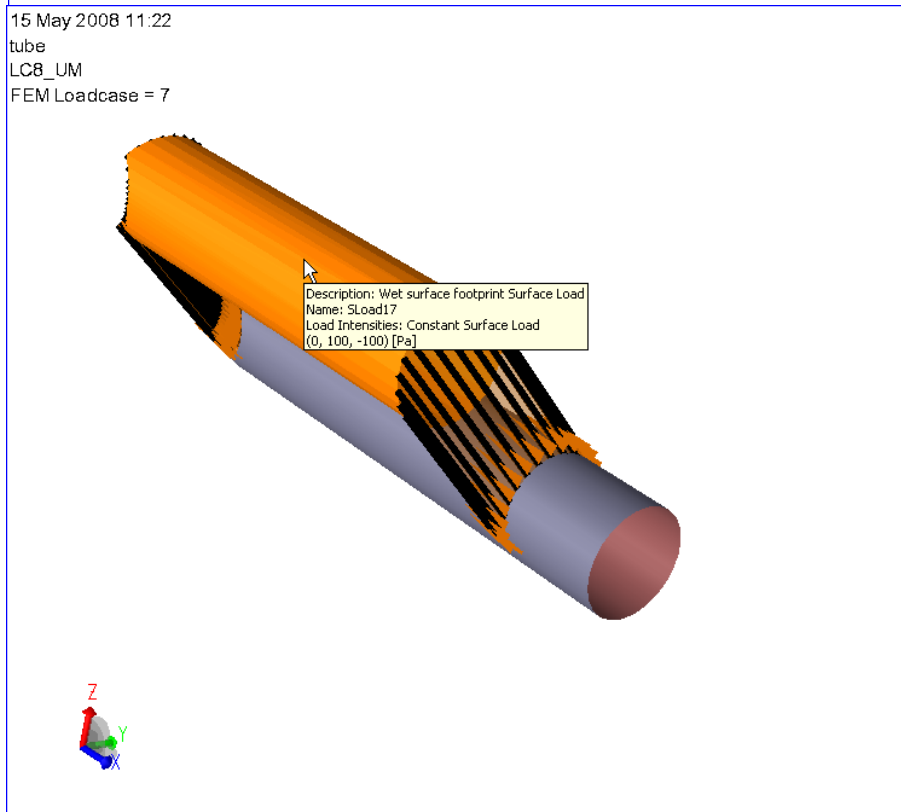
Intensities
Constant Component Load (Component2d)
Fx: 0 Pa [Pa]
Fy: 100 Pa [Pa]
Fz: 0 Pa [Pa]



15 May 2008 11:22
tube
LC8_UM
FEM Loadcase = 7

Footprints
Wet Surface
Select a wet surface to apply loads on
WS_Inner

Intensities
Constant Component Load (Component2d)
Fx: 0 Pa [Pa]
Fy: 100 Pa [Pa]
Fz: -100 Pa [Pa]



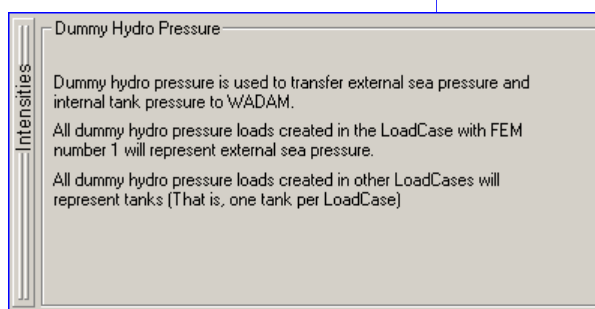
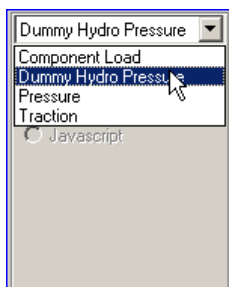
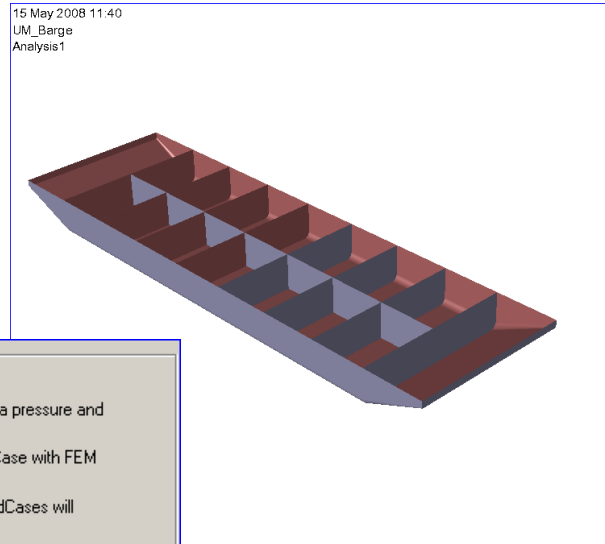
A constant pressure with components in y and z directions.

4.3.3.4 Transfer hydro pressure data to HydroD

This load type is a special case to determine which parts of the structure that shall receive pressure loads in a hydrodynamic analysis by using HydroD (stability, Wadam or Wasim analysis). This means that these special loads define the outer hull and the internal tanks (or compartments). In most cases the compartment information is defined as shown in the next Section, but it is also possible to do it manually as described later in this section.

A barge is used to explain how to create loadcases with dummy hydro pressure. The top deck has been removed for visibility.

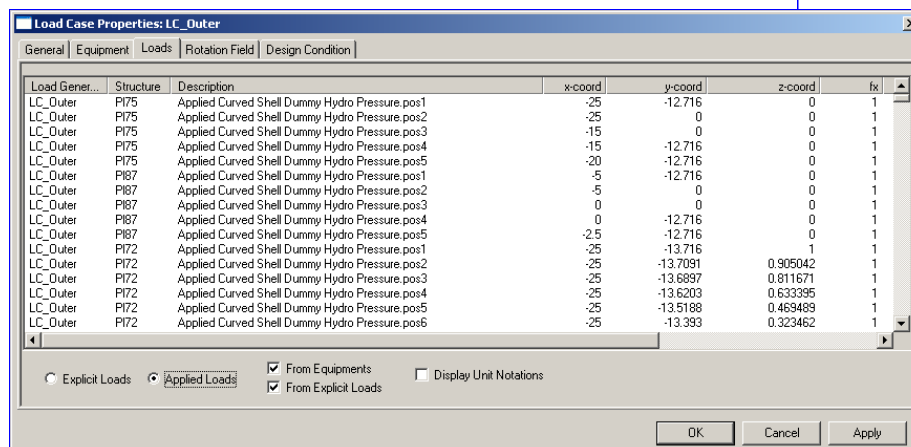
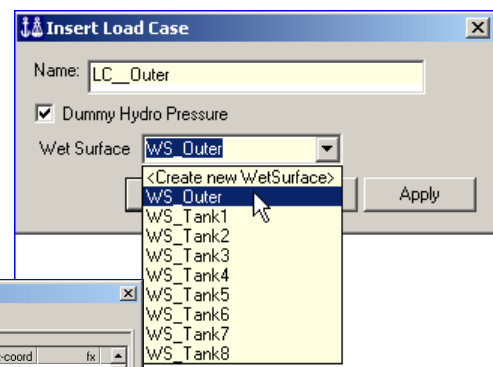
The dummy hydro pressure loads are generated by using the option Dummy Hydro Pressure.



No load intensities are defined. GeniE will make the necessary data to be used by HydroD. Note that the external wet surface is defined in both the panel model and the structural model (when applicable) while the internal compartments are defined in the structural model only.

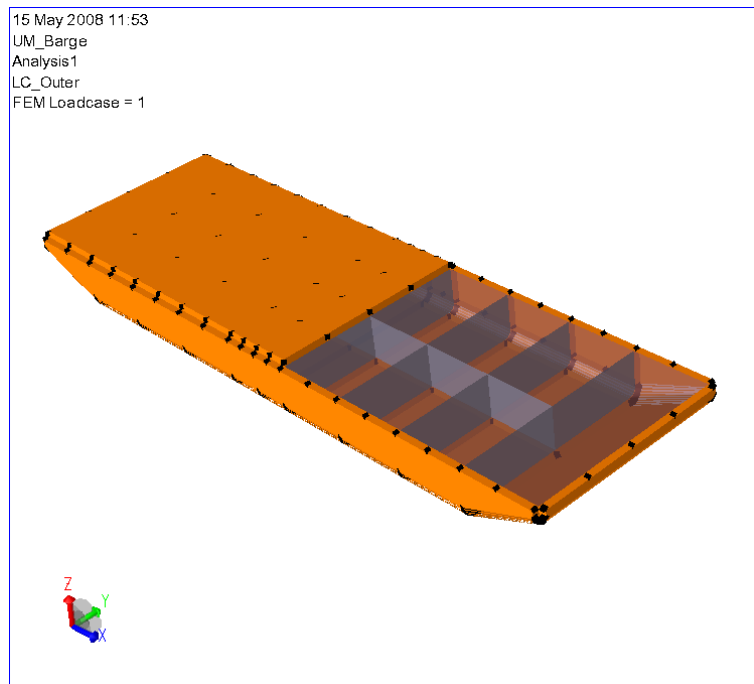
Define the external surfaces exposed to sea pressure

It is required that the FEM load case number is 1 (i.e. the first loadcase). When defining such a loadcase it is also required to tick off the option for the dummy hydro pressure. In this case, there is one wet surface (WS_Outer) that includes all the outer part of the hull including the deck. The dummy hydro pressure can be verified from e.g. the load case property dialogue box:



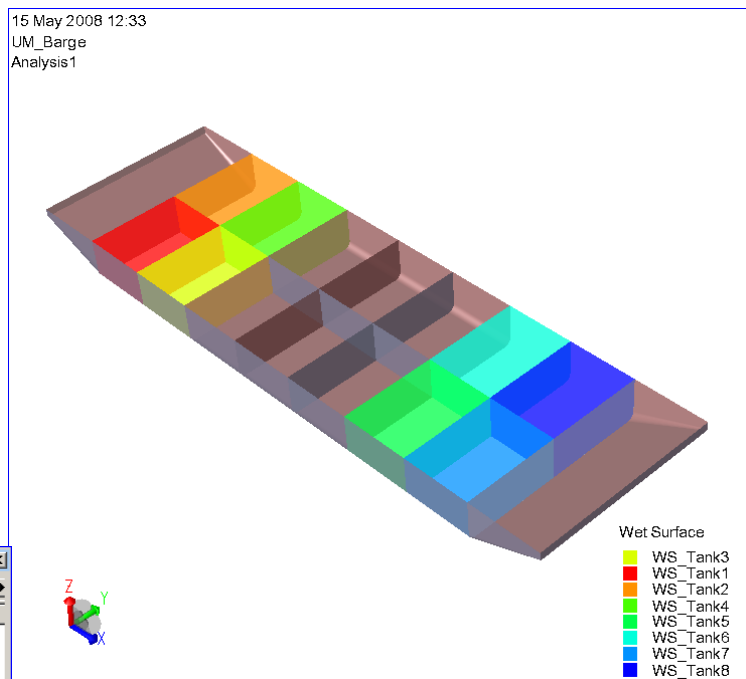
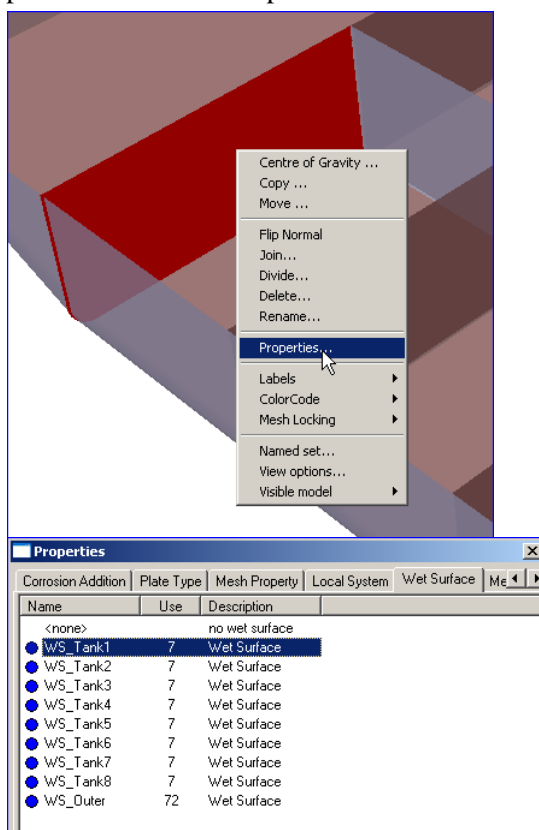
A graphic visualisation (after the applied loads have been generated) is shown to the right. Some of the loads on the deck have been removed for visibility.

When running HydroD the stability and hydrodynamic analyses will include the parts defined by the dummy hydro pressure. Other parts will be neglected.



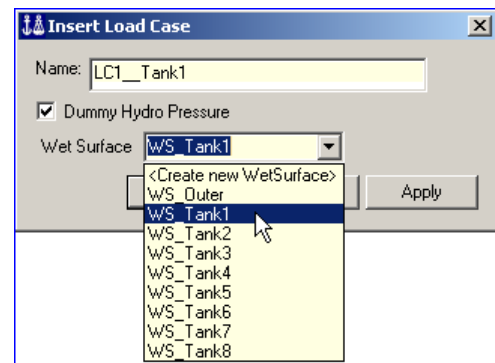
Manually define a tank (compartment) for use in HydroD

The principle for creating tank definitions for use in HydroD is the same as when defining the external sea pressure: Define wet surfaces, assign these to relevant plates and define a load case including dummy hydro pressure. For a rectangular cubic compartment there needs to be at least 6 surfaces connected to the same wet surface property (remember to assign the wet surface to the right side of the plate). A compartment is generated by referencing one unique wet surface. In this case there are 8 wet surfaces applied to several plates to define 8 compartments.



The load case describing the tank can now be created by including the relevant wet surface. In this case the load case LC1_Tank1 will define the dummy hydro pressure for the wet surface WS_Tank1. HydroD will interpret this as a compartment that can be used in ballasting and subsequent pressure computations.

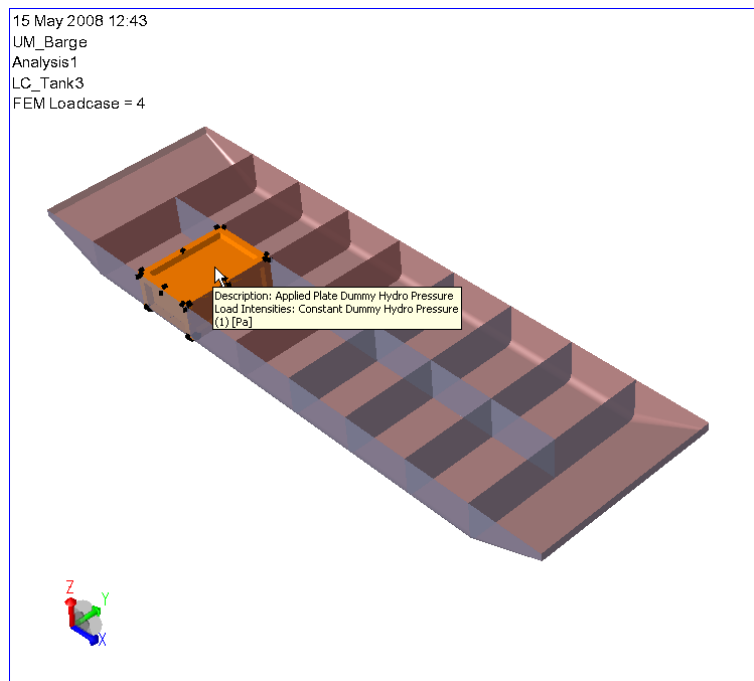
The FE loadcase numbers for compartments must start with number 2, i.e. sequential to the loadcase defining the external sea pressure. There must be one loadcase per compartment.



The dummy hydro pressure loads may be verified from e.g. the load case property dialogue box or graphically as shown to the right (in this case the pressure loads for LC_Tank3 based on the wet surface WS_Tank3). The deck plate is removed for visibility.

The manual definition of compartments requires that you assign a wet surface to the right side of all plates (or shells) needed to define a closed volume (tank/compartment).

The next Section shows a much quicker approach using automatic generated compartments.



4.4 Compartment loads

Compartments are used to define loads and information for use in:

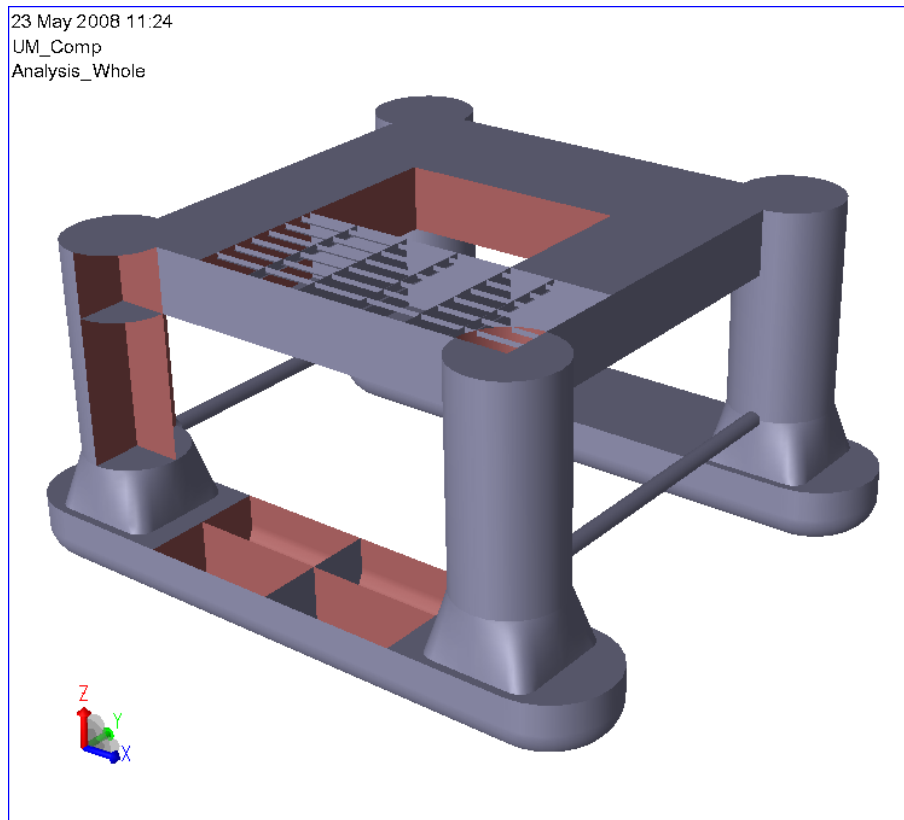
- Analyses where the loads are explicitly defined by the user. The loads may be user specific or in accordance with rules. The analyses may be carried out from GeniE using a pre-defined activity (see next Chapter) including Sestra.
- Analyses where the loads are automatically applied according to the CSR Bulk rules. This requires that the concept model (including compartment information) is exported to Nauticus Hull FEA template. In addition to compartment loads the boundary conditions, loads to the outer hull and corrosion addition are also automatically applied to the model. The analyses may be carried out from GeniE using a pre-defined activity (see next Chapter) including Sestra
- Hydrostatic, stability and hydrodynamic analyses where the compartments and the outer wet surface are used to define the panel model (buoyancy and sea pressure) and the compartments subjected to various content and filling degrees. The hydrodynamic analyses and the subsequent structural analyses are carried out using Brix Explorer configured for Sesam (including HydroD, Wadam/Wasim and Sestra).
- Structural analysis where corrosion addition is included. Corrosion addition may be added to the compartments so that the additions apply to the plates/shells and relevant beams/stiffeners.

4.4.1 Create compartments

Compartments are defined by using the pull down menu **Insert/Compartment/Compartment Manager**. Compartments will now automatically be generated where there is a closed volume. Compartments are automatically adjusted when the model is updated, typically by moving a bulkhead or adding information like watertight/non-watertight bulkheads.

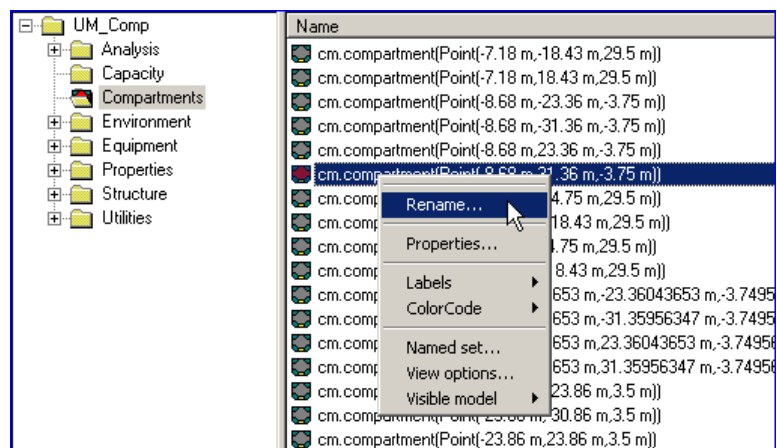
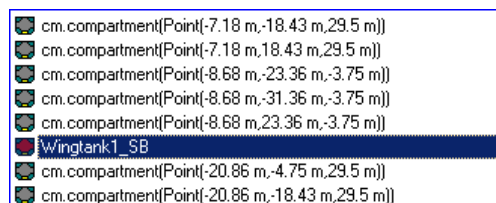
Some examples are shown in the following focusing names, visualisation, modifications, dummy structural plates (in case you have an open compartment to be loaded), the use of non-watertight bulkheads and corrosion addition.

The model to the right is used as a reference case. Notice that some of the plates have been removed to illustrate that there are closed volumes in the concept model.

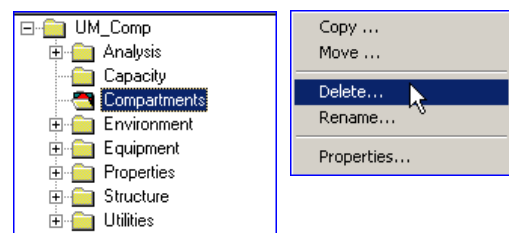


4.4.1.1 Visualise and rename

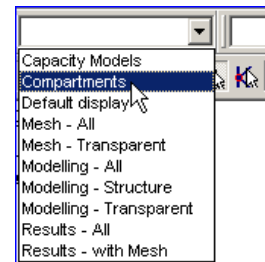
The names of the compartments relate to the position of the compartments centre. You may rename the compartment name to typically *Wingtank1_SB*.



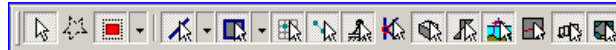
If you delete the compartment manager and make a new compartment manager all compartment names are reset to program default.



Compartments can be viewed by using one of the pre-defined views, typically the *Compartments* view.

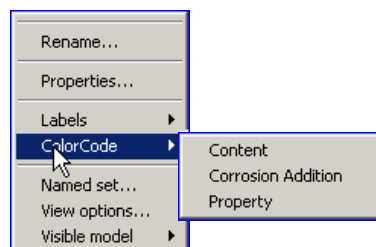
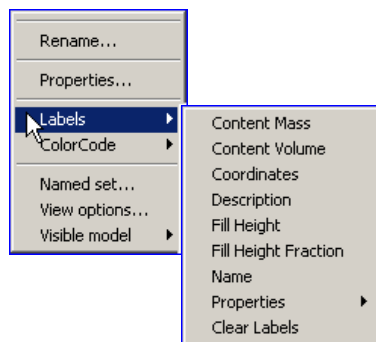


You can modify the settings of this view or make your own views as explained in Volume 1 of the User Manual.

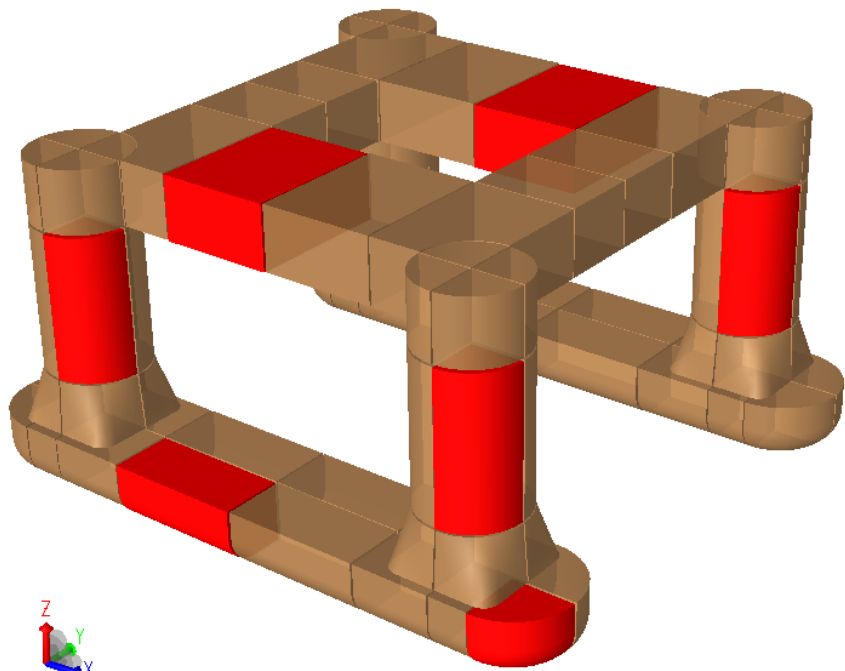


The picture to the right shows the compartments automatically generated when defining a compartment manager from the pulldown menu *Insert/Compartment/Compartment Manager*. There are seven compartments that are highlighted.

You may add label information and do colour coding as shown below.

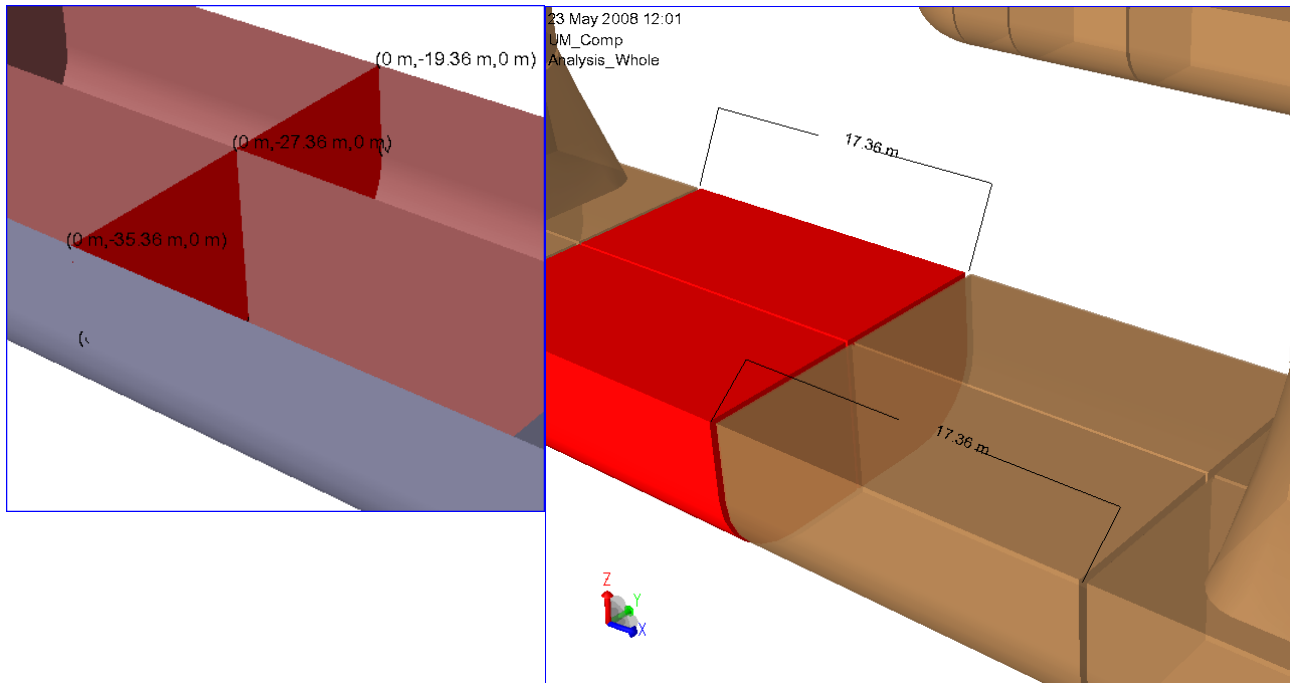


23 May 2008 11:43
UM_Comp
Analysis_Whole

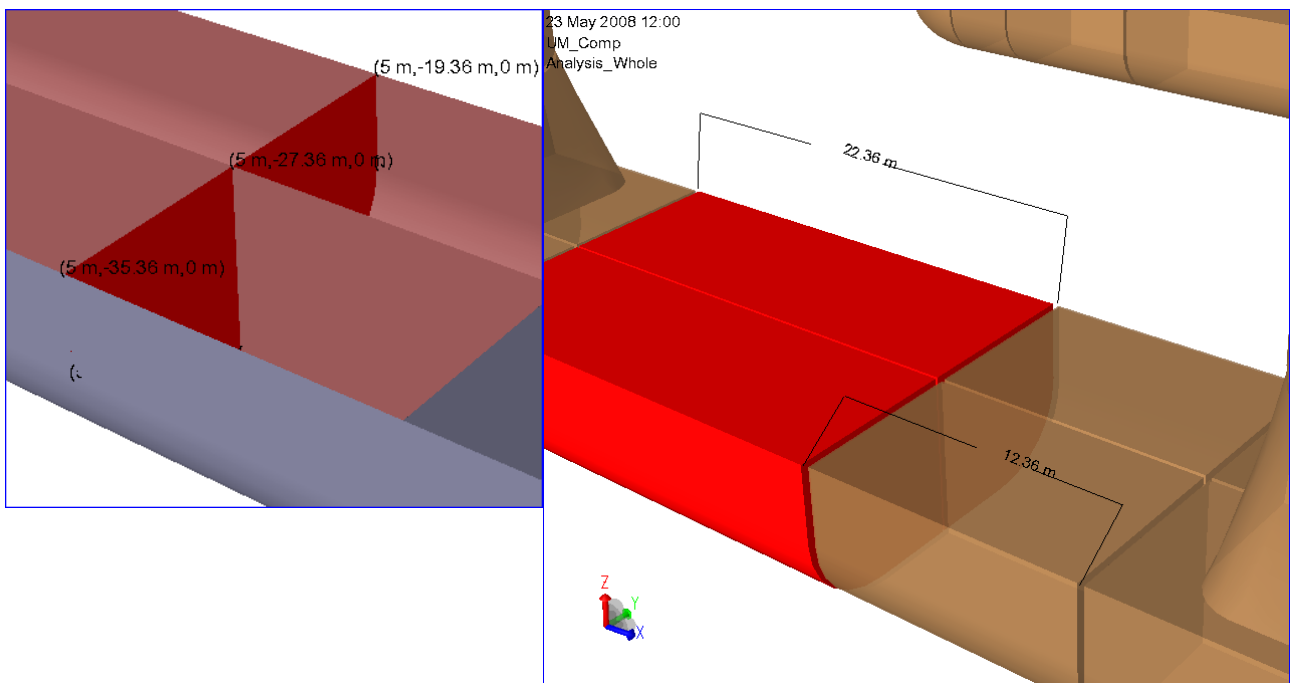


4.4.1.2 Modify structure

When you modify the structure layout (e.g. move a bulkhead) the compartments and its content or loads are automatically adjusted. The example below shows the effect of moving a bulkhead 5 meters in horizontal direction.



Compartment layout after moving the bulkhead 5 meters:

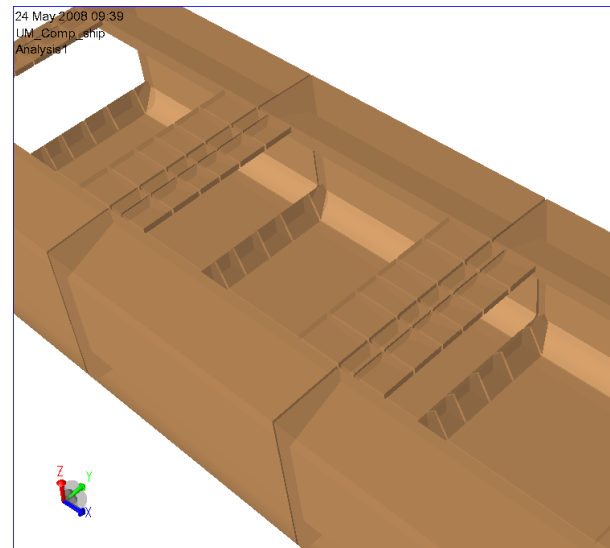
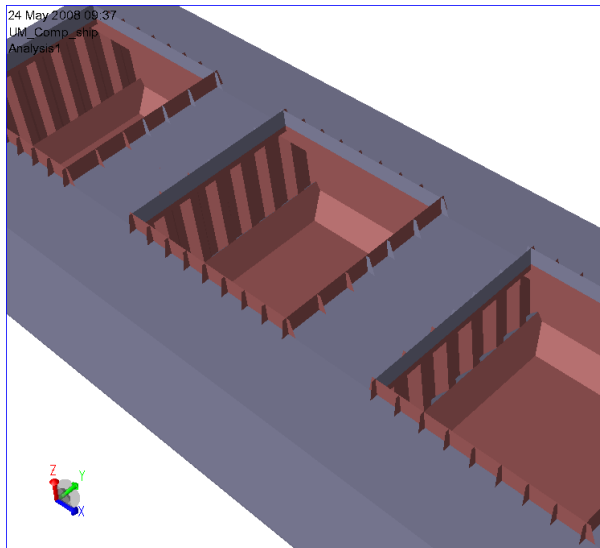


As can be seen the lengths of the compartments are adjusted in accordance with the structural change.

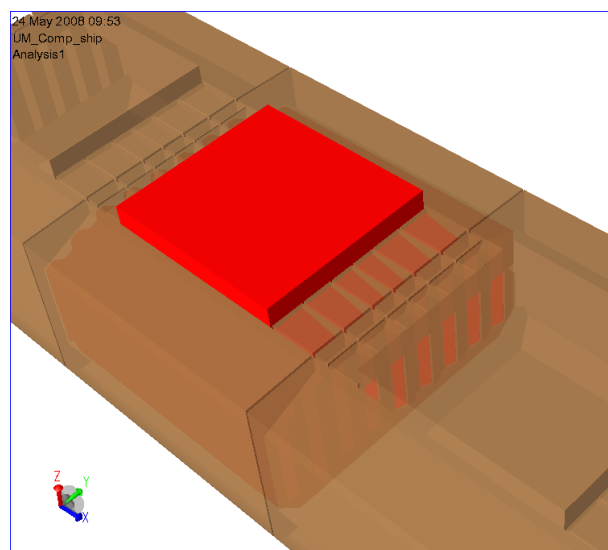
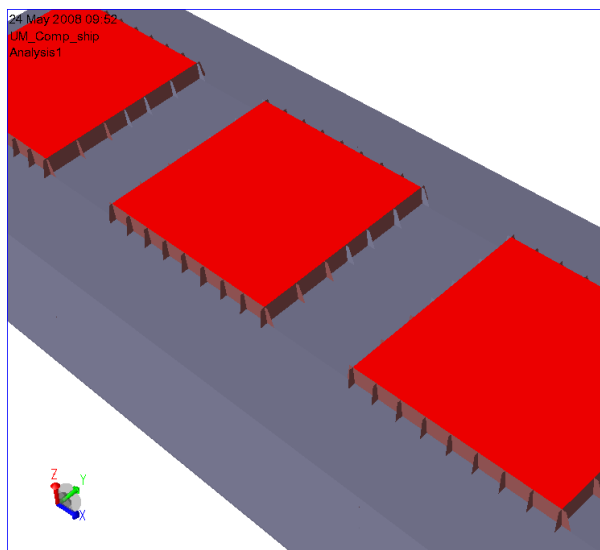
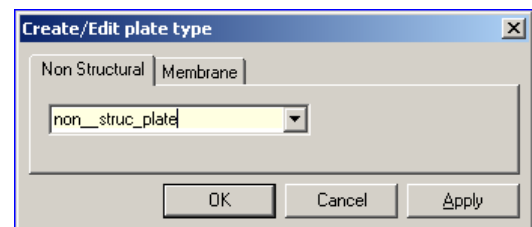
4.4.1.3 Open compartments and non-structural plates

A compartment is defined when there is a closed volume. However, there are cases where compartments can be defined when there is an opening in the volume. An example of such may be bulk or container ships where the top of the compartment is open. To close the volume without adding extra material and stiffness a non-structural plate may be used. The purpose of a non-structural plate is to close the opening and as such they are not adding stiffness or material. A non-structural plate can not carry any loads.

In the example below a typical 3 cargo hold model of a bulkship is shown. There are openings in the three centre compartments; hence no compartments are generated.



To close the opening a non-structural plate needs to be inserted. Make a non-structural property from **Edit/Properties/Plate Type/Non-Structural**. You can also do this from the browser. Remember to apply the property to the plates in question.

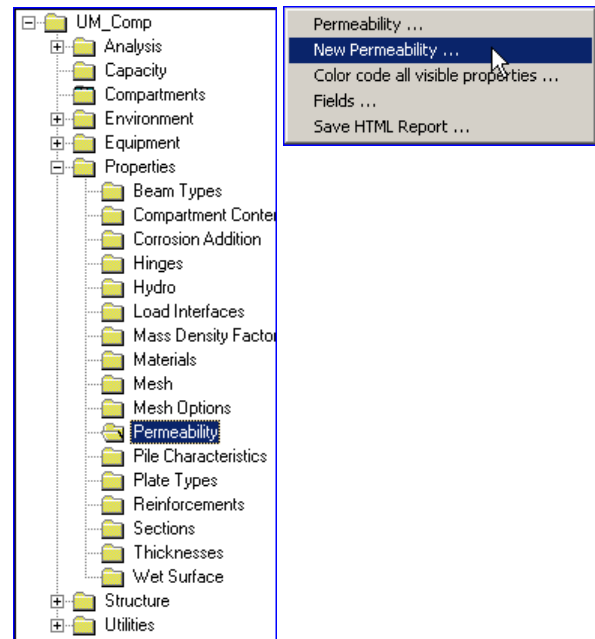
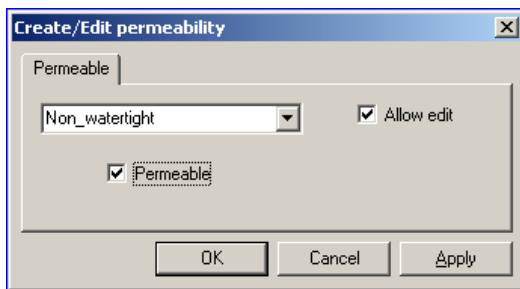


Non-structural plates are inserted at the top of the hatch coamings and compartments are automatically generated. The mid compartment is highlighted above.

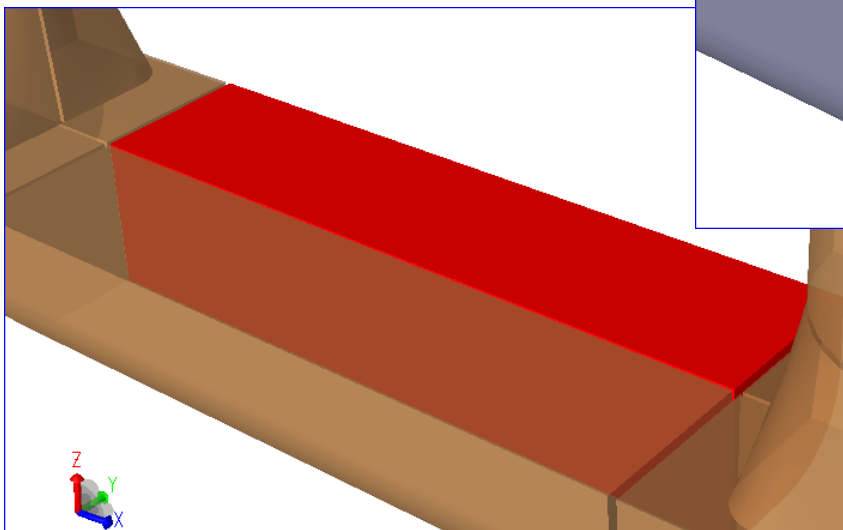
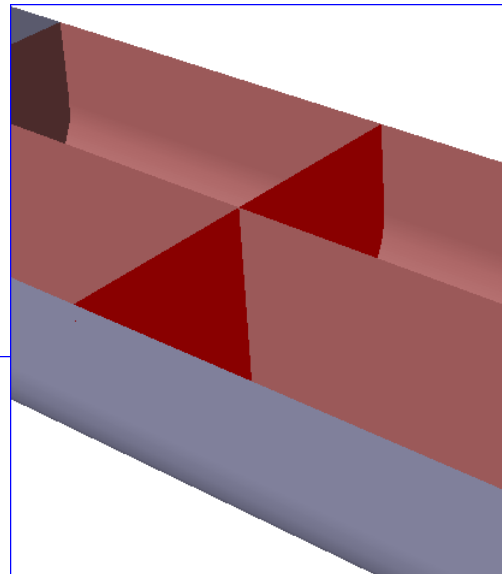
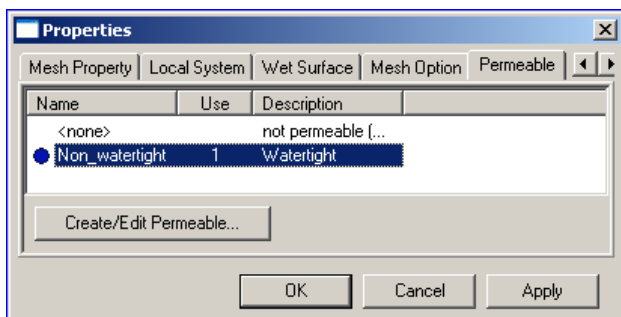
4.4.1.4 Non-watertight bulkheads

A compartment may have internal walls that are non-watertight. In case you want to include the effect of such walls (stiffness, material, loads) they may be classified as non-watertight. In this case the actual plate is disregarded when defining the outer boundaries of a compartment. When calculating loads etc. the non-watertight plate will receive loads and corrosion addition (see following Sections).

Prior to assigning a plate to non-watertight it is necessary to define a non-watertight property from **Edit|Properties|Permeable** or from the browser. Remember to *not* tick-off “Permeable” to ensure non-watertight conditions.



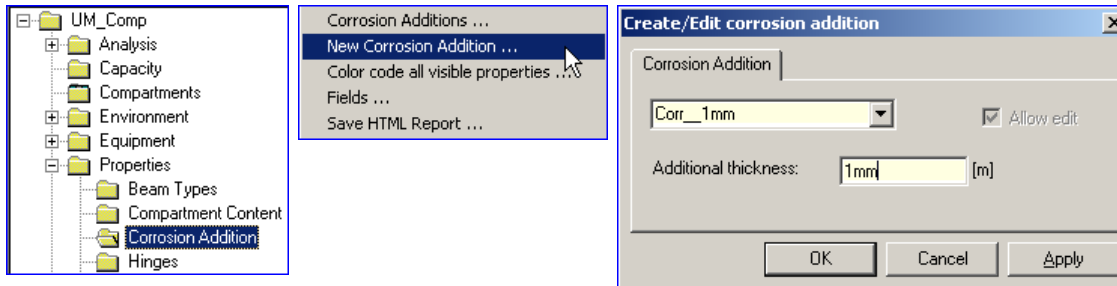
In the example below the highlighted plate is assigned the *Non_watertight* property. As can be seen the compartment is now bounded by the outer parts of the closed volumes.



4.4.1.5 Corrosion addition

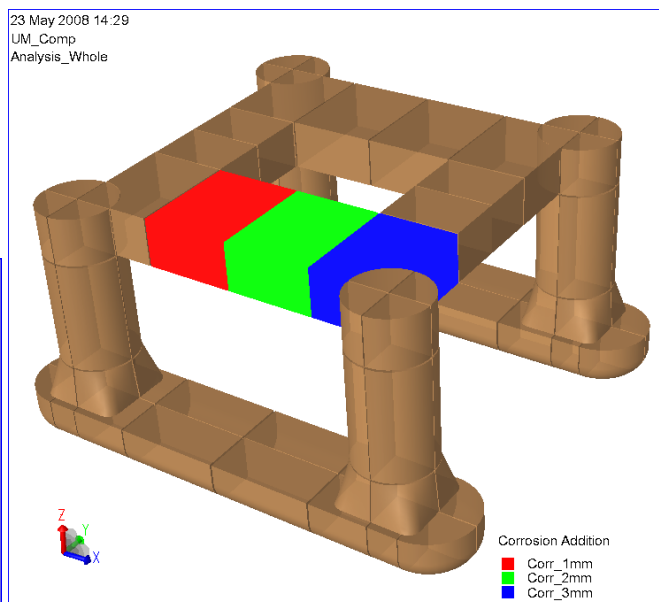
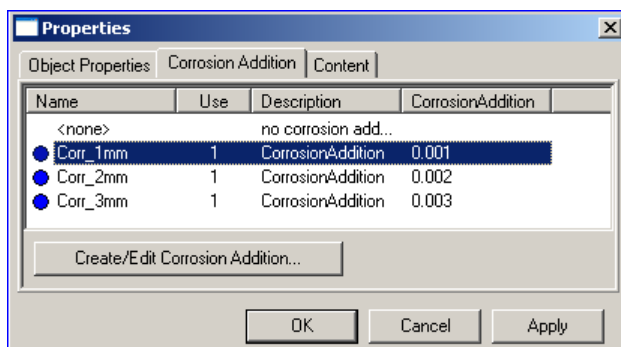
Corrosion addition may be added to a compartment. In this case the corrosion addition is automatically applied to the right side of the plates (pointing towards the inside of the compartment) as well as to the stiffeners inside a compartment. It is necessary that the stiffeners are assigned eccentricities – this is used to detect if a stiffener is inside a compartment or not. The corrosion addition is applied according to Section 6.2.1.5.

In the example below corrosion addition has been applied to three compartments at deck level.

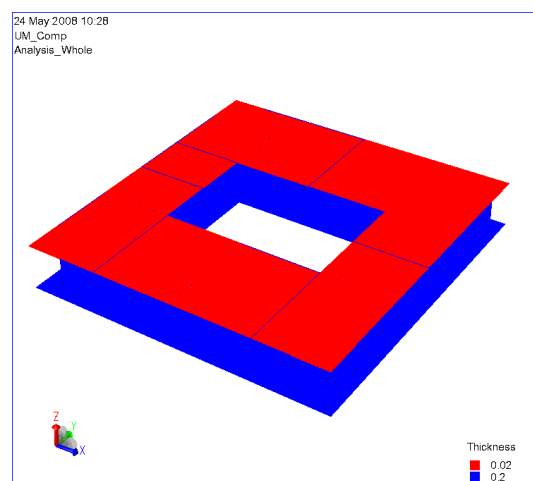
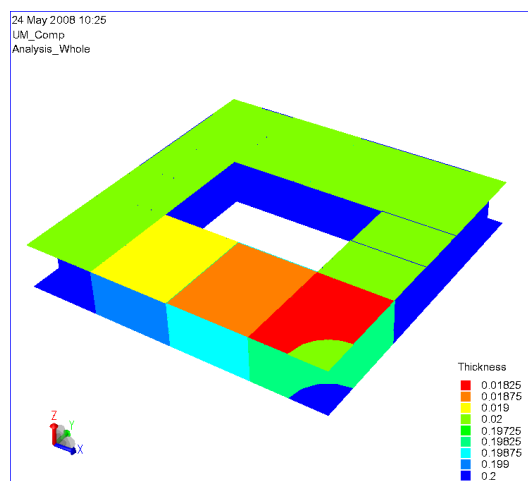


Corrosion addition applied as shown to the compartments.

Select a compartment, **RMB** and click *Properties* to apply the corrosion addition.



When you include the corrosion addition in the mesh generation (*Edit/Rules/Meshing* and activate *Scantlings to msNetCSR_Bulk*) the effective mesh plate thicknesses (and stiffener properties) are reduced.



Effective mesh plate thicknesses with and without corrosion additions are shown above (see also Chapter 6).

4.4.2 Design load based analysis – manual load application

There are two ways of generating surface loads in a compartment. The first option is to fill the compartment with a liquid or solid content and specify the filling. Pressure loads are now automatically defined based on the acceleration (normally gravity) for the actual loadcase. The surface loads are applied according to the footprint of the content, i.e. the load may be applied to parts of a plate. The pressure load is always normal to the plates.

Furthermore, it is not necessary to manually create a mesh line along the surface load footprint as GeniE automatically will create finite element loads. The quality of the load generation depends on the mesh density – for global analysis it is sufficient to have a coarse mesh while for refined analysis a finer (smaller) mesh densities are required.

The other option is to apply surface loads to the entire compartment or to parts of it by selecting plates belonging to the compartment. There are a number of surface load types that can be applied, for details see previous Section *Surface loads on shells*.

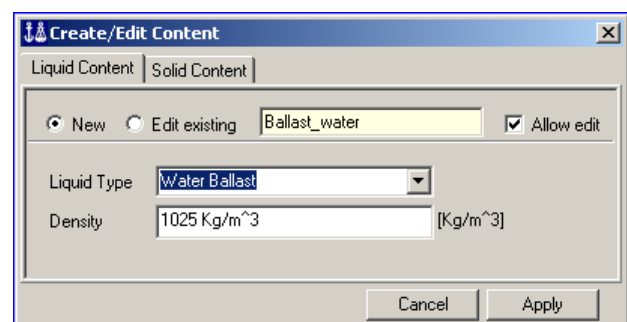
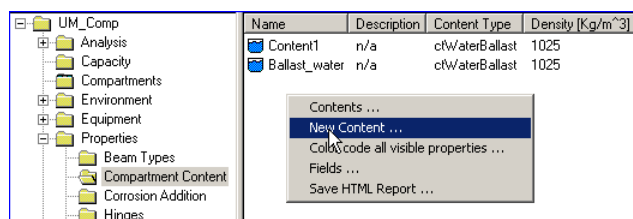
Common for both options is that a loadcase must be active to define the loads. Some examples are shown in the following to explain how to define surface loads based on compartments.

4.4.2.1 Content

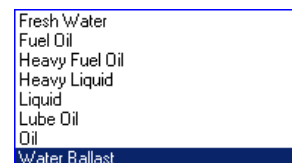
Once you add a content to your compartment, the local system will be changed from "Global" to "Replace Z with fill height", unless you have explicitly modified the local system of the compartment prior to assigning content. This applies both to solid and liquid content.

4.4.2.2 Liquid content

Liquid contents may be defined from **Edit/Properties/Content/Liquid Content** or from the browser.

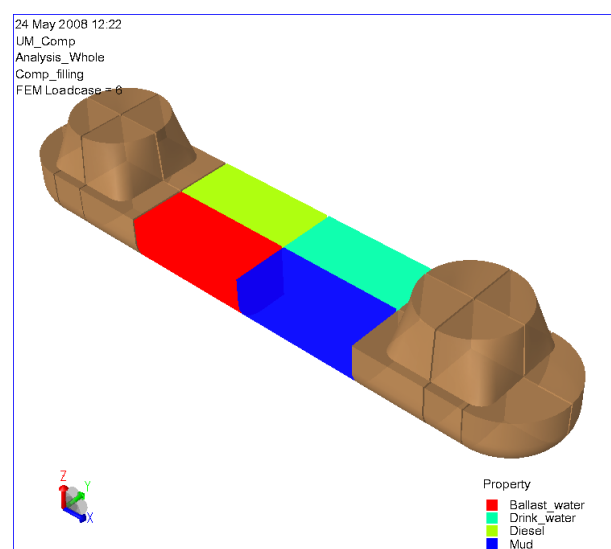


It is possible to specify a number of liquid types, and depending on the type different colours are used when visualising the content. In the example below four liquid contents have been defined and applied to four compartments.



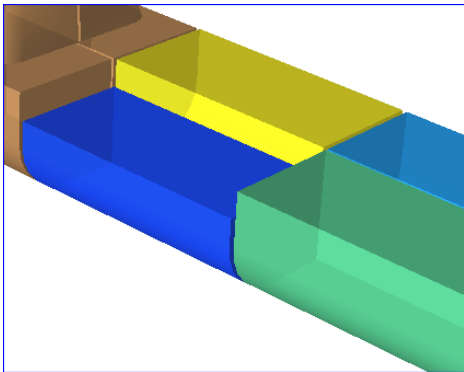
The colours used are:

- Blue: Water ballast
- Light blue: Fresh Water
- Yellow: Fuel Oil, Heavy Fuel Oil, Lube Oil, Oil
- Turquoise: Heavy Liquid, Liquid



The default filling of a compartment is 100%, i.e. the compartment is full. The filling degree is modified by selecting a compartment, **RMB**, select Fill Height and specify the fill height fraction, the fill height, the cargo volume or the cargo mass.

The example to the right shows that the fill height fraction is set to 0.75. Observe that the volume of the compartment is also calculated, in this case 986 m³.



Properties

Object Properties | Content | Fill Height | Compartment Loads

☐ Reference point

☒ Replace Z with fill height

☒ Fill height fraction (h/H) 0.75

☐ Fill height (h) 7.5 m [m]

☐ Cargo Volume (V) 985.7094603 m³ [m³]

☐ Cargo Mass (M) 1010352.197 Kg [Kg]

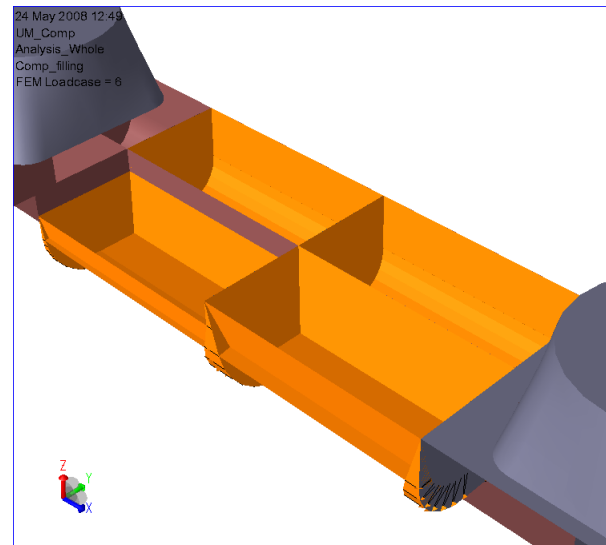
OK Cancel Apply

The above fill height fraction is applied to the compartment filled with water ballast. The compartment view (to the right) is automatically updated.

The surface loads are automatically generated when you make a finite element model or you run an analysis. It is also possible to manually generate the surface loads by selecting the loadcase, **RMB** and *Generate Applied Loads*. The loads may be visualised and verified from e.g. the loadcase property dialogue.

As can be seen the compartments are filled to the top except for one. Furthermore, there are no pressure acting on the top plate; an example of pressurized compartments is shown in the following.

Some of the plates are removed for better visibility of the loads.



Load Case Properties: Comp_filling

General | Equipment | Loads | Rotation Field | Design Condition

Description	x-coord	y-coord	z-coord	fx
Applied Compartment Pressure.pos1	0 m	-35.36 m	-3.75 m	55162.4 Pa
Applied Compartment Pressure.pos2	17.36 m	-35.36 m	-3.75 m	55162.4 Pa
Applied Compartment Pressure.pos3	17.36 m	-35.36 m	0 m	0 Pa
Applied Compartment Pressure.pos4	0 m	-35.36 m	0 m	0 Pa
Applied Compartment Pressure.pos5	0 m	-35.36 m	-1.875 m	27581.2 Pa
Applied Compartment Pressure.pos1	0 m	-31.36 m	-7.5 m	110325 Pa
Applied Compartment Pressure.pos2	17.36 m	-31.36 m	-7.5 m	110325 Pa
Applied Compartment Pressure.pos3	17.36 m	-27.36 m	-7.5 m	110325 Pa
Applied Compartment Pressure.pos4	0 m	-27.36 m	-7.5 m	110325 Pa
Applied Compartment Pressure.pos1	0 m	-35.36 m	-3.75 m	55162.4 Pa
Applied Compartment Pressure.pos2	17.36 m	-35.36 m	-3.75 m	55162.4 Pa
Applied Compartment Pressure.pos3	17.36 m	-35.2237 m	-4.72057 m	69439.5 Pa
Applied Compartment Pressure.pos4	17.36 m	-34.8241 m	-5.625 m	82743.6 Pa
Applied Compartment Pressure.pos5	17.36 m	-34.1884 m	-6.40165 m	94168.1 Pa
Applied Compartment Pressure.pos6	17.36 m	-33.36 m	-6.9976 m	102934 Pa
Applied Compartment Pressure.pos7	17.36 m	-32.3953 m	-7.37222 m	108445 Pa

☐ Explicit Loads ☒ Applied Loads

☒ From Equipments ☒ Display Unit Notations

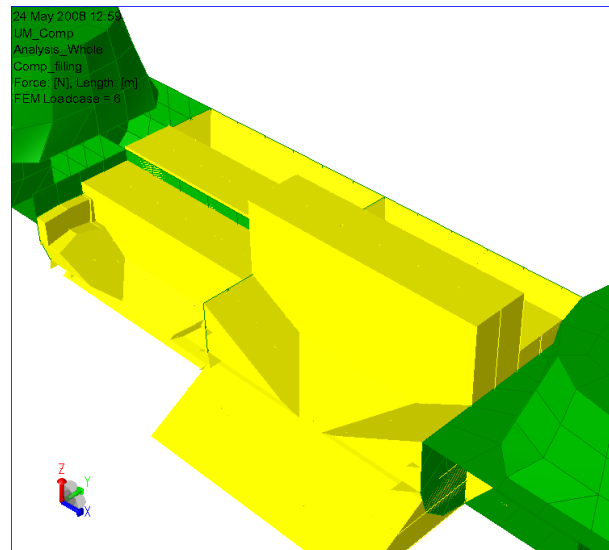
☒ From Explicit Loads

OK Cancel Apply

The generated loads in the finite element model are shown to the right. As can be seen the compartment with 75% filling receives less load. Furthermore, the loads are different for the other compartments because the contents have different material densities. The higher density, the larger the load becomes.

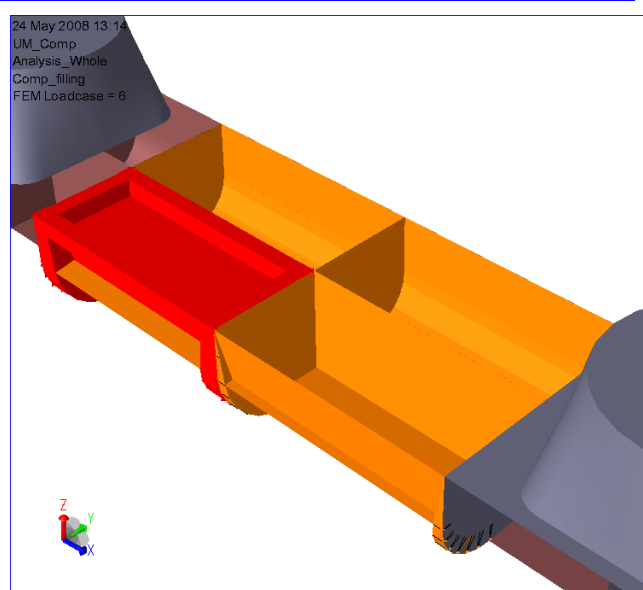
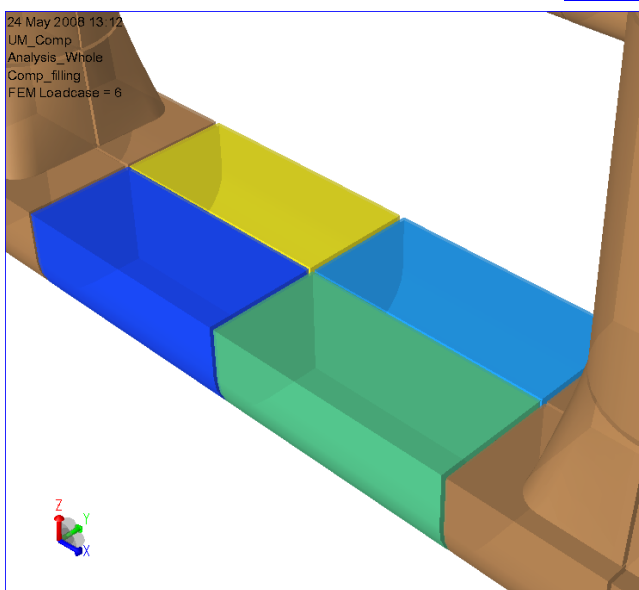
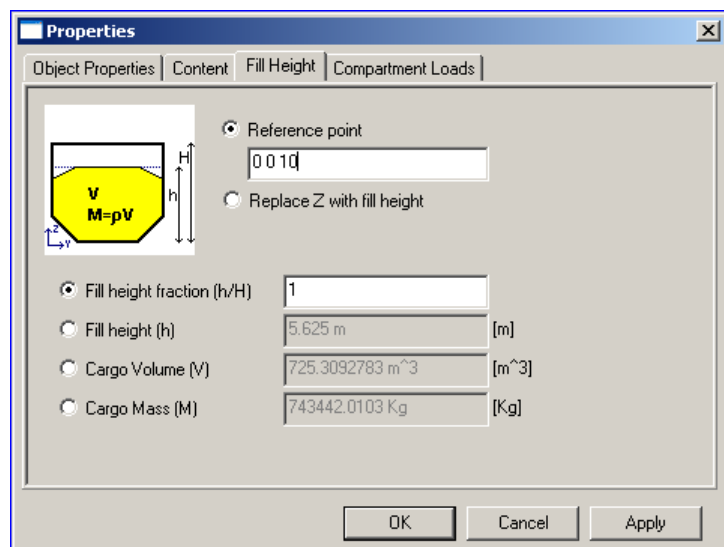
Some of the plates are removed for visibility and hence not all acting pressure loads are shown.

For more details on how to create a finite element model see Chapter 6.



In case you have a compartment that is pressurised (e.g. when filling in a pipe connected to the compartment) typically you can specify the top position of the filling by modifying the reference point for computing the loads.

In the example to the right the reference point is set to (0m, 0m, 10m) using a fill height fraction of 1. This fill height is applied to the water ballast compartment and as can be seen, there is now also a pressure load acting upwards on the top plate of the compartment.



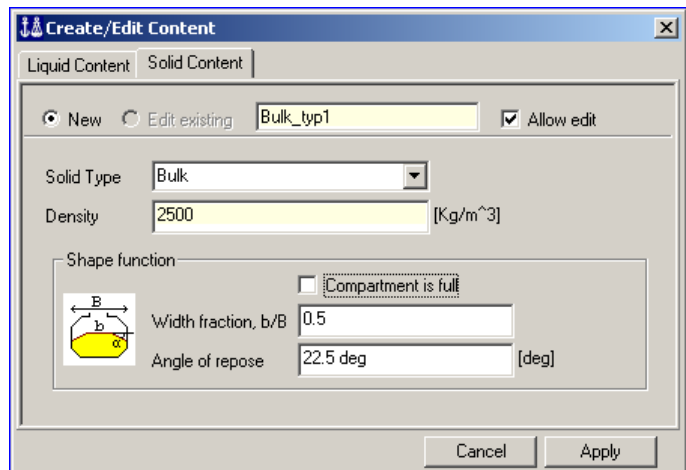
The pressure acting on the top plate of the compartment is highlighted above.

4.4.2.3 Solid content

Solid content may be applied to the compartments like for liquid content. In addition it is possible to specify the shape function of the top of the content. It is possible to define three types of solid contents.

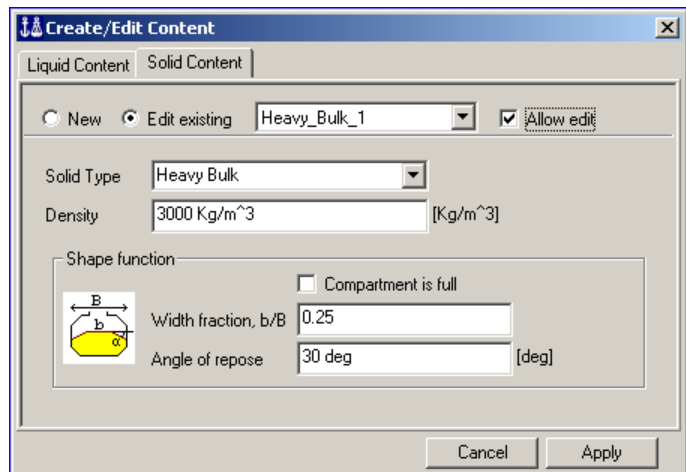
In this case solid type *Bulk* is selected. The shape function is using the default values.

Compartment colour type is brown.



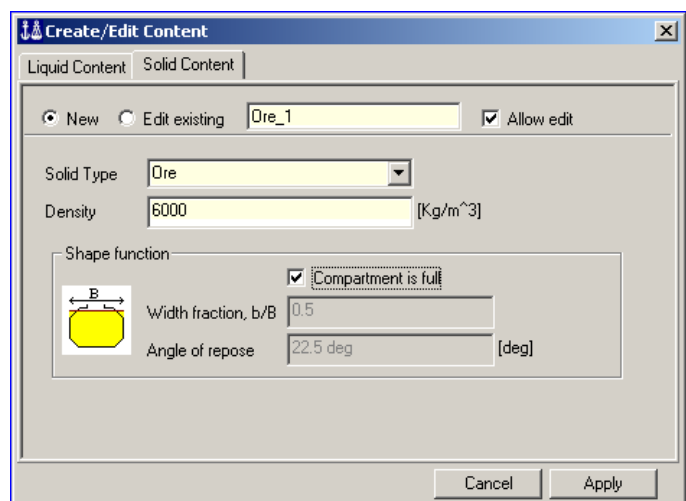
The dialogue box to the right shows a content defined using *Heavy Bulk* and a modified shape function.

Compartment colour type is black.

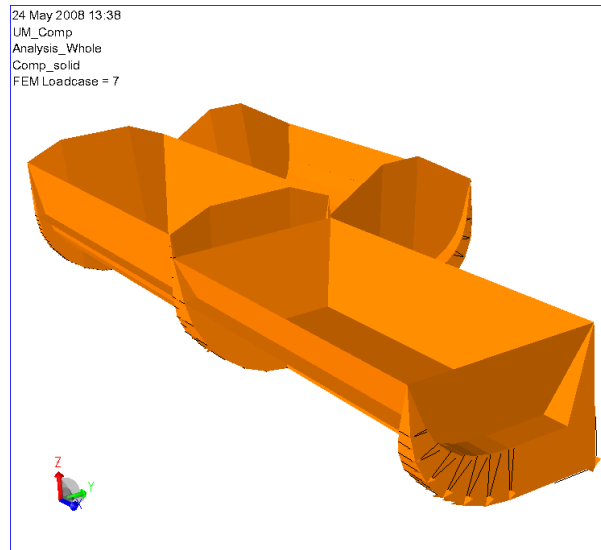
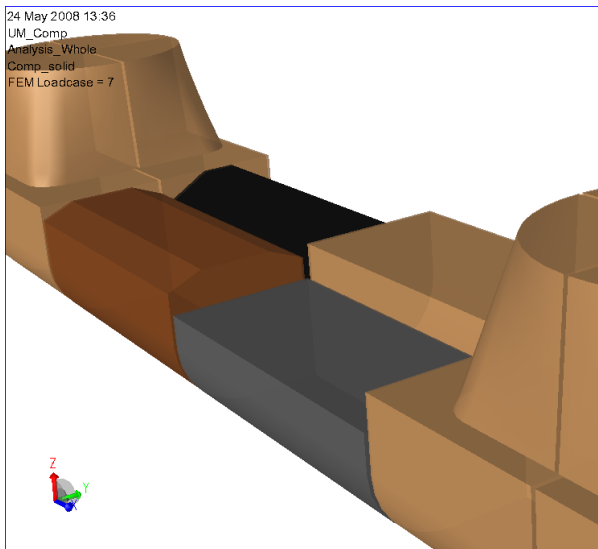


The third option is *Ore*. In this case the compartment is assumed to be full. In this case the top shape function is disregarded.

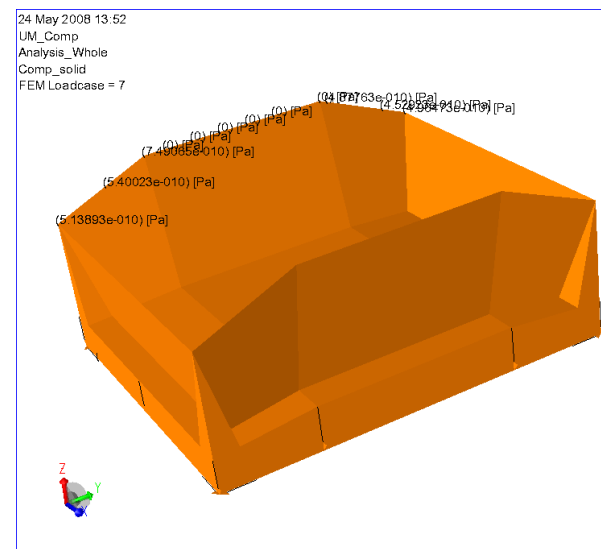
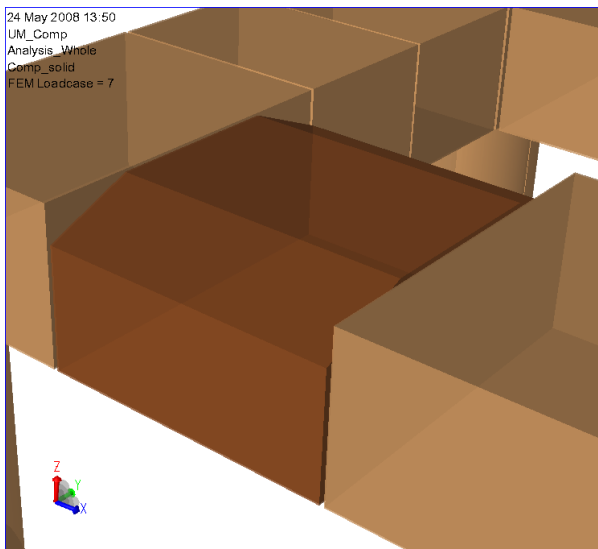
Compartment colour type is grey.



The content types are applied to three compartments. The compartment containing ore has a filling degree of 75%. Observe that the top of the ore has a horizontal distribution since the shape function has been disabled.

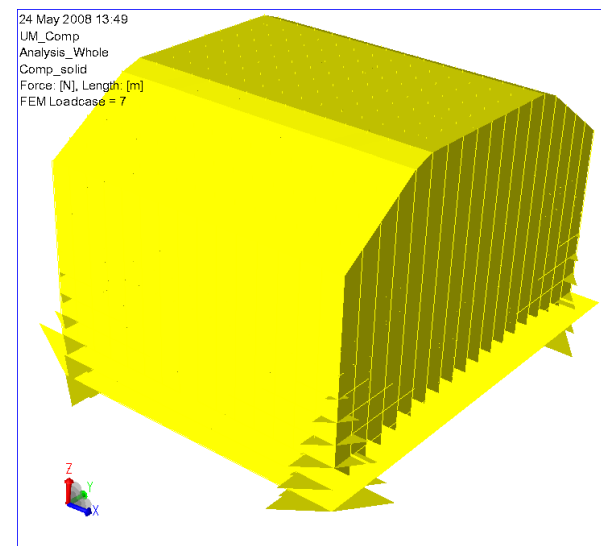


When applying a solid content with a shape function to a regular compartment it is easier to understand how the loads are generated and how they are mapped to a finite element model.



In this case the finite element discretization is good enough to make a representative finite element load application. If the finite element mesh is much coarser, the loads may not be identical to the applied loads. In such cases the total load sums are maintained, but not representative for local detailed analysis.

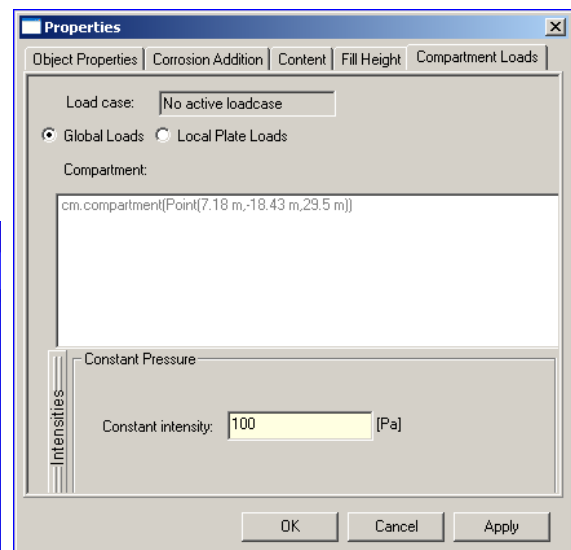
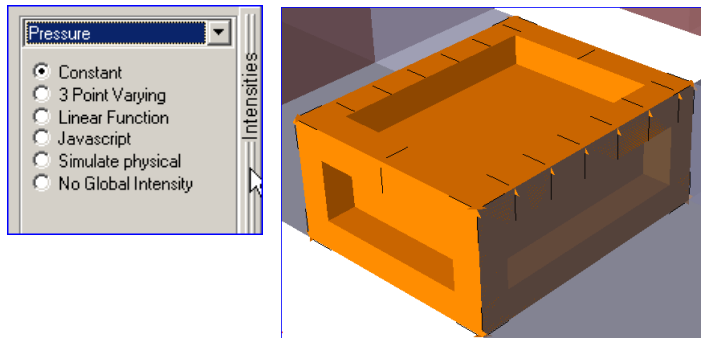
The finite element loads are shown to the right.



4.4.2.4 Manually defined compartment loads

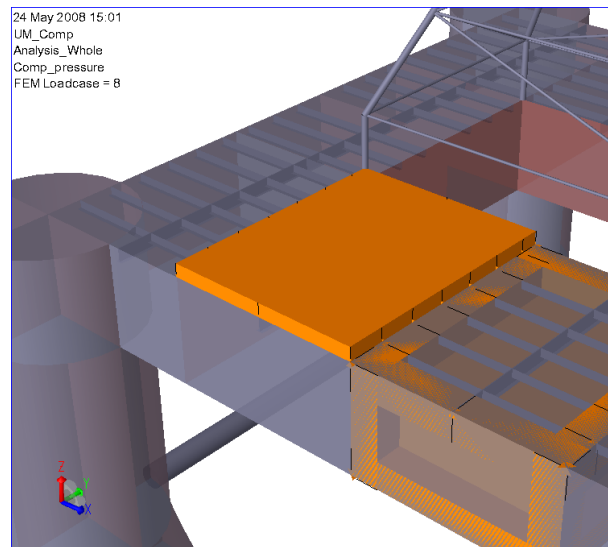
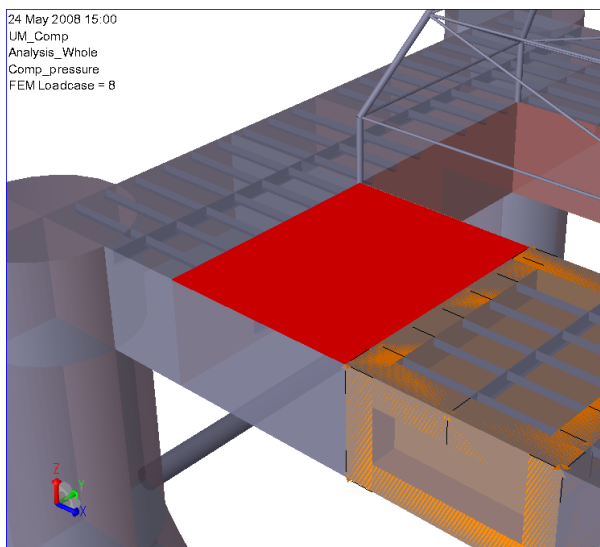
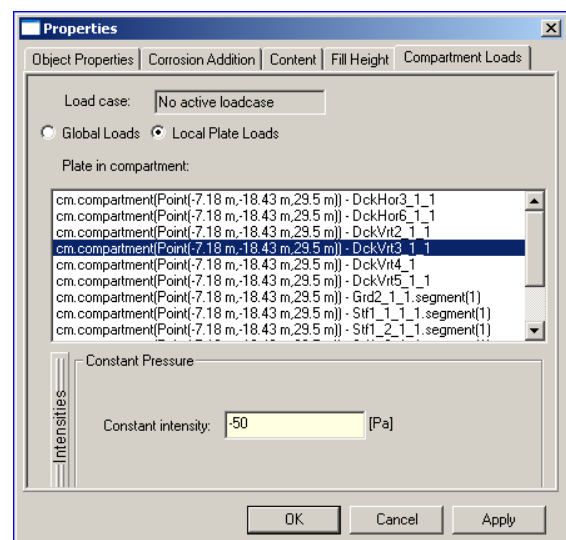
It is also possible to specify pressure loads manually to a compartment. Select a compartment, **RMB** and use the option *Compartment Loads*.

The load definitions are the same as documented in the previous Section.



In the example above the constant pressure load of 100 Pa is applied to the inside of all plates belonging to the selected compartment.

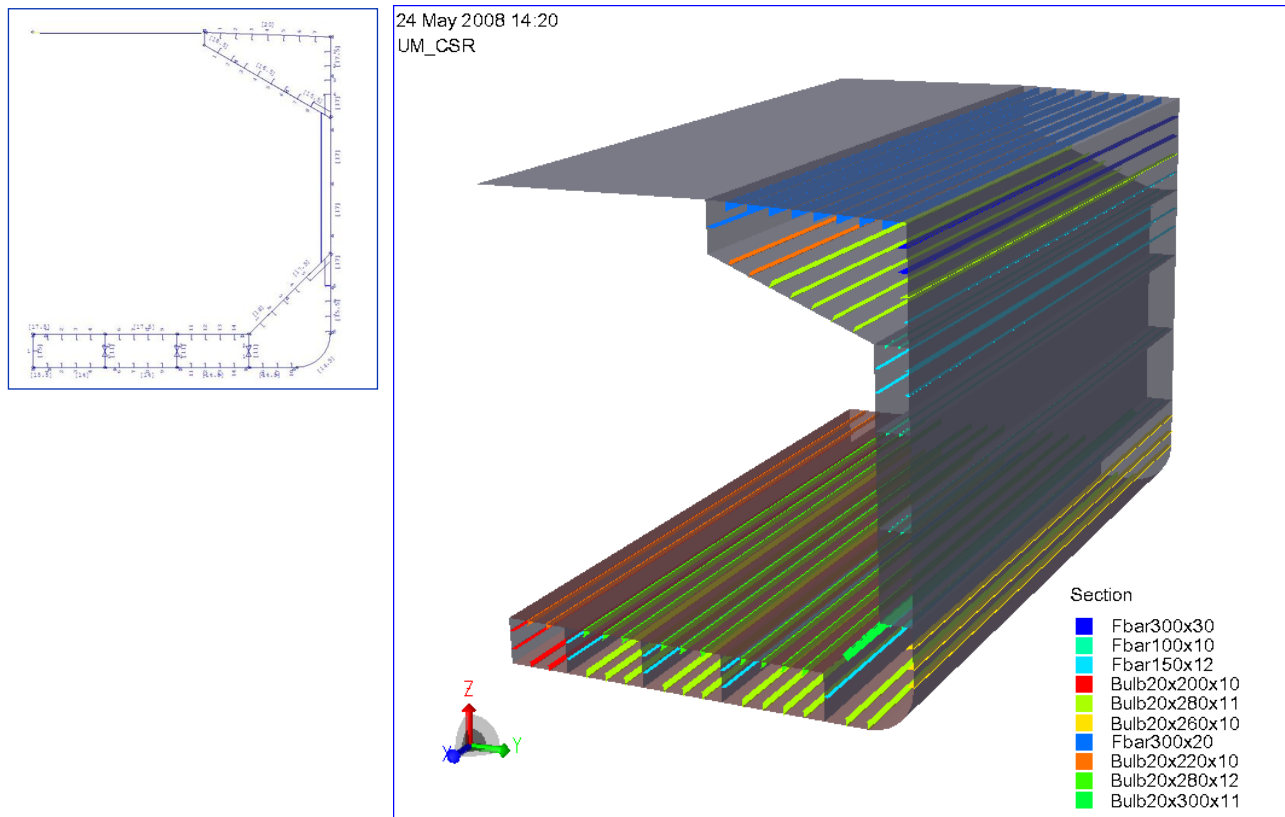
When applying loads to some of the compartment plates, the option “Local Plate Loads” is used. Select a compartment, **RMB** and use the option *Compartment Loads* and click *Local Plate Loads*. To better see the plates you are selecting you should now switch to a view where you see the plates (for example the view “Modelling Transparent”).



The pressure is acting towards the centre of the compartment since it is negative.

4.4.3 Design load based analysis – rule based load application

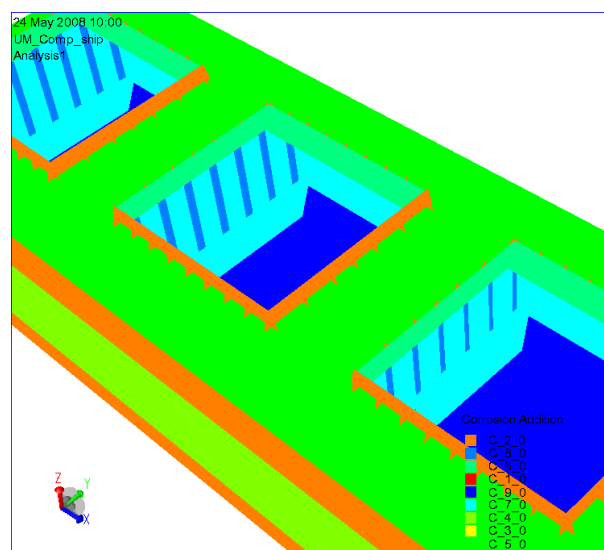
When using Nauticus Hull to define section scantlings based on the CSR rules for bulk ships it is possible to export the longitudinal material data (plates and stiffeners) for re-use in GeniE. The pictures below show a typical mid-ship cross section as defined in Nauticus Hull Section Scantlings and the similar model in GeniE after importing the relevant data. The stiffeners are highlighted and as can be seen they have the right properties as well as eccentricities.



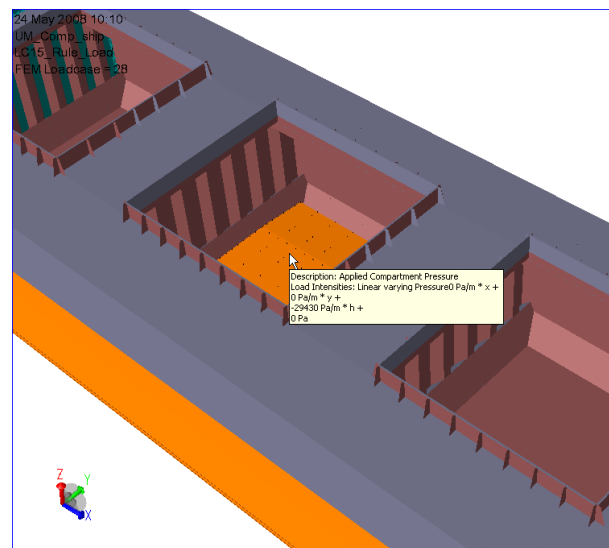
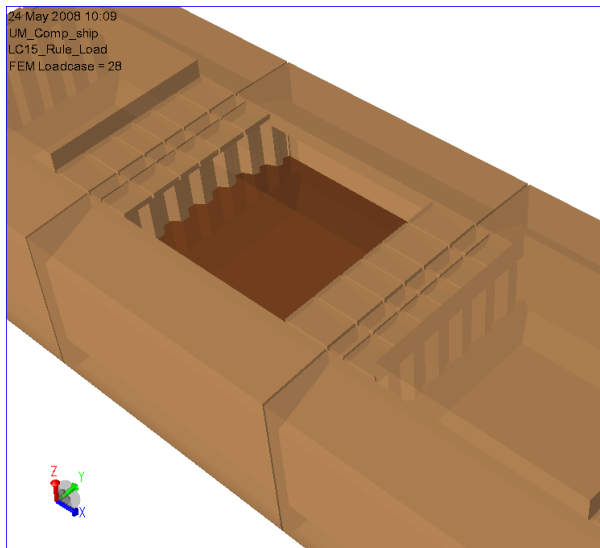
When the remaining parts of the structure has been modelled in GeniE the compartment definitions can be exported to Nauticus Hull for a subsequent definition of corrosion addition, boundary conditions as well as loadcases and content according to the CSR Rules. This is an automatic process with no manual definitions involved. The rule based information can be imported to GeniE for an automatic update of the model so that it contains corrosion addition, boundary conditions and a number of loadcases to automatically satisfy the requirements by the CSR.

In the following some examples are shown to illustrate how automatic generated properties according to the CSR for bulk ships are applied to a GeniE model.

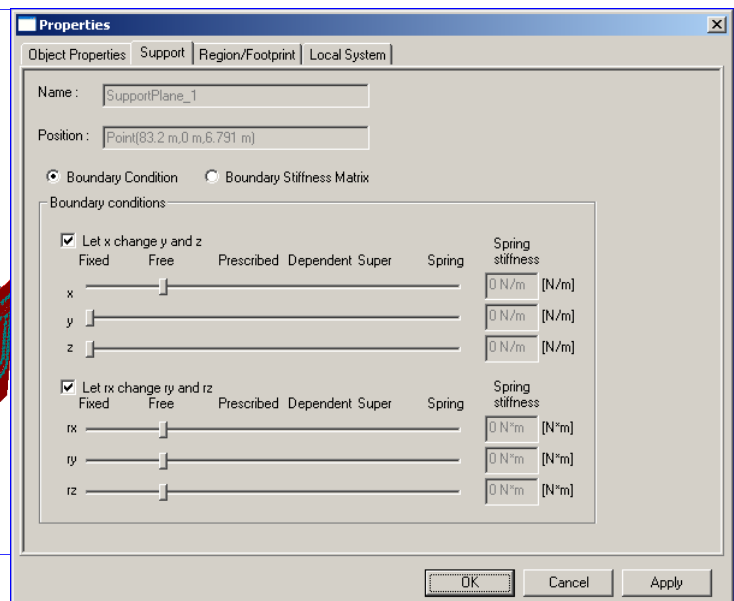
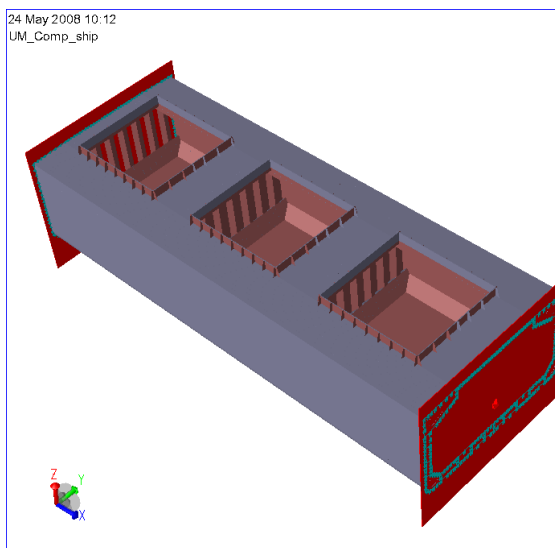
Corrosion addition automatically applied to the model (plates and stiffeners)



One of the loadcases automatically generated is half filling of the centre cargo hold with bulk. The compartment content and the generated pressure load are shown below (pressure inside the compartment as well as hydrostatic pressure on the outer hull).



A 3 cargo hold analysis must have proper boundary conditions to simulate the inclusion of the complete structure. This is automatically done according to the rules and can be visualised and documented as shown below (the boundary conditions are different for both ends).



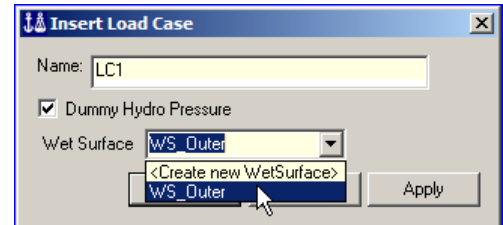
All export from and to GeniE in this case is based on XML files.

4.4.4 Direct analysis - transfer compartment data to HydroD

HydroD can re-use compartment information to set up its internal volumes (tanks or compartments) that can be used in hydrostatic, stability and hydrodynamic analyses (including ballasting) to define additional mass as well as to receive accelerations and pressures from wave load analysis.

HydroD requires that the first loadcase defines the outer wetted surface, while the following loadcases are used to define the compartments. Note that the compartments are defined in the model referred to as the structural model in HydroD.

The example to the right shows how to define the first load case used to define the wetted surface (see also previous chapters) while the remaining loadcases are used to define compartments for use in HydroD.

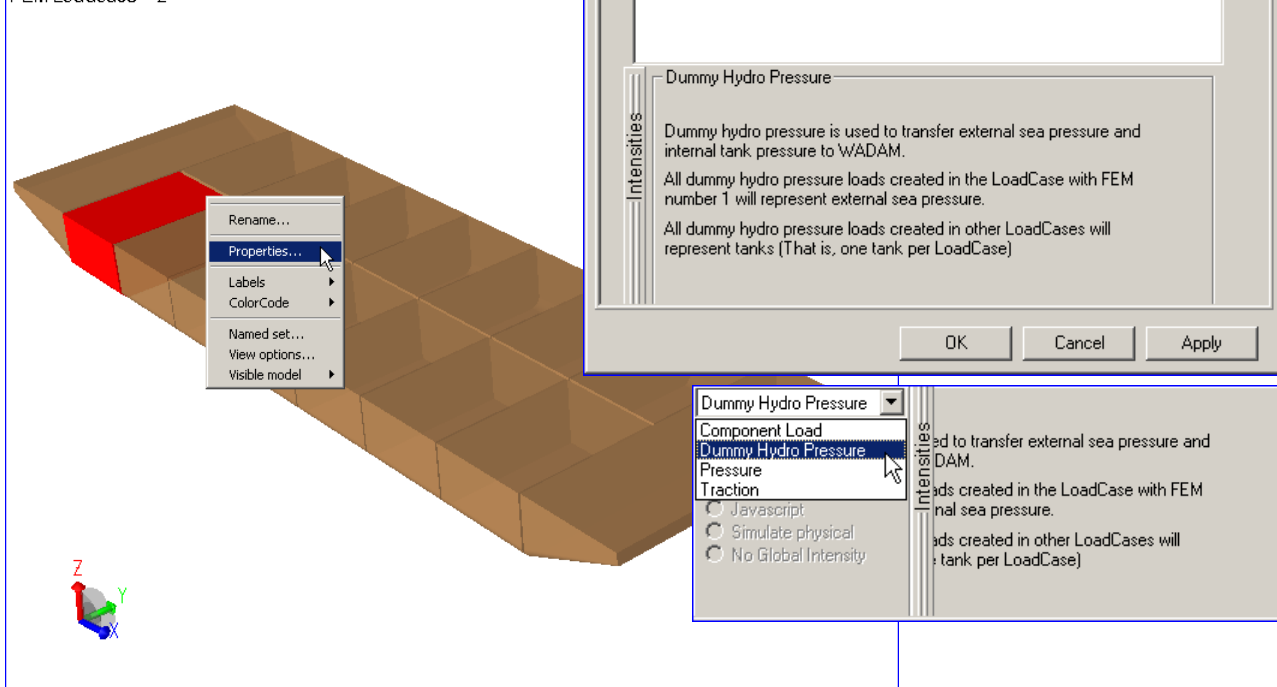


To add a compartment with so-called “dummy hydro pressure” for use in HydroD:

- Define a load case
- Select a compartment, **RMB** and select *Compartment Loads*.
- Use intensity “Dummy Hydro Pressure”
- Make a load case per compartment that shall be part of HydroD analysis

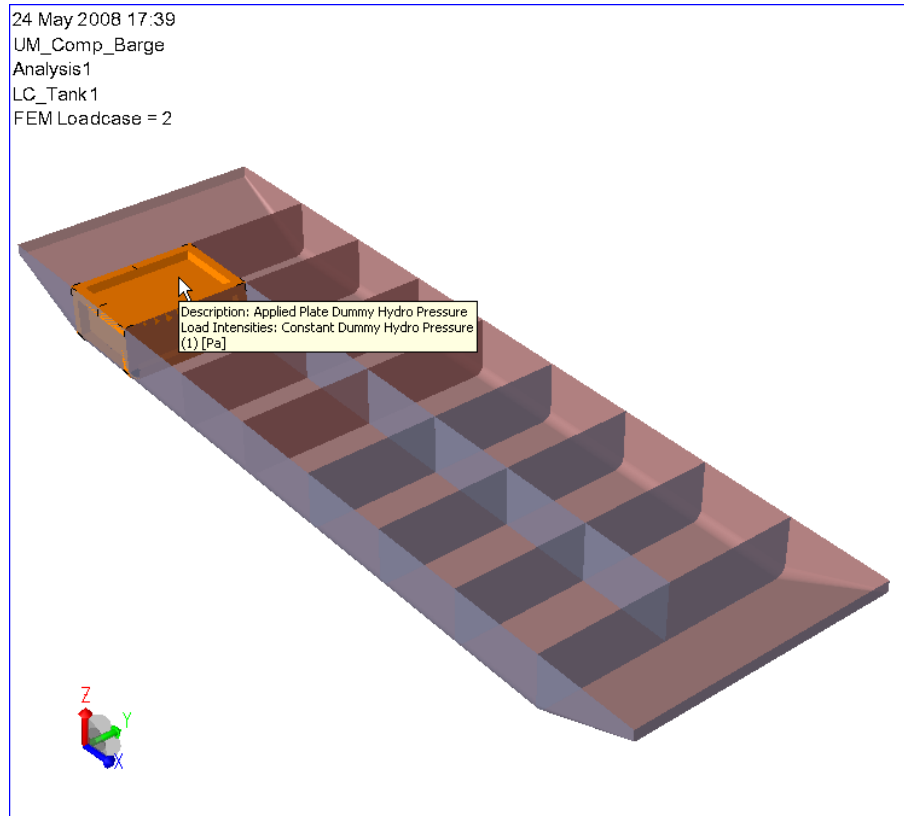
The same barge as in previous sections is used to show how compartments can be used to define the dummy hydro pressure much quicker than a manual approach where the dummy hydro pressure is applied for each plate to simulate a compartment.

24 May 2008 17:33
UM_Comp_Barge
Analysis1
LC_Tank1
FEM Loadcase = 2

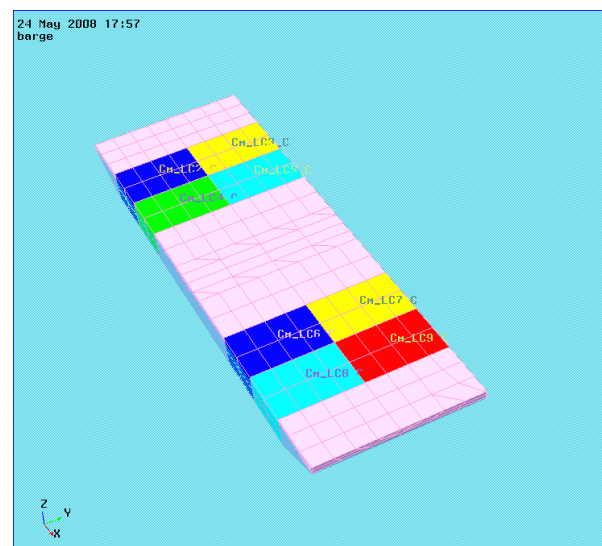
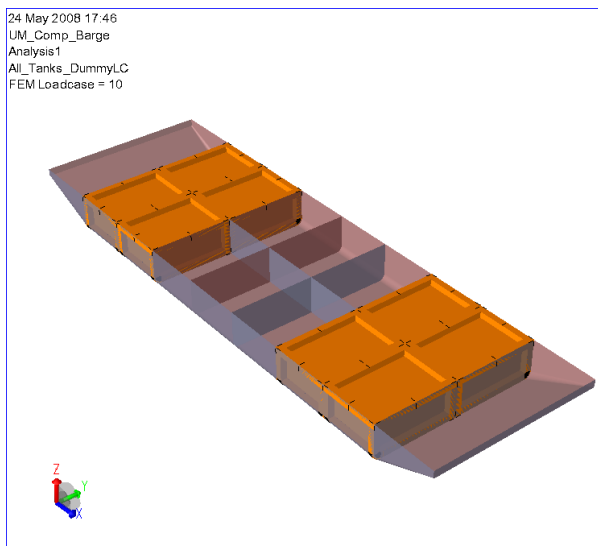


The applied loads from the compartment are now constant pressures for each of the compartment plates, intensity 1 Pa (as required by HydroD).

The actual pressure will be calculated in a subsequent wave load analysis performed by HydroD.



In this case eight loadcases are defined; each one containing dummy hydro pressure for one compartment. The picture shows the eight loadcases in one view (load combination) and how they appear in HydroD. Notice that it is not possible to import a load combination in HydroD. For more details, please consult the HydroD user manual.



4.5 Equipment loads

Equipments may be defined and re-used in various positions in different load cases. When there is an intersection between the equipment footprint and a beam(s), loads or masses will be created depending on your requirements. The equipment can as such be used to make a model intended for linear structural analysis or dynamic analyses like structural or hydrodynamic.

Updated loads or mass positions are automatically created when the equipment is moved or modified, provided there is interface between the footprint and a beam(s). You can also use load calculation rules (or load interfaces) to determine which beams that shall be loaded. There is always equilibrium between the equipment loads and the applied beam loads. Typically, if there is a horizontal acceleration a force couple will be automatically generated in addition to a shear force.

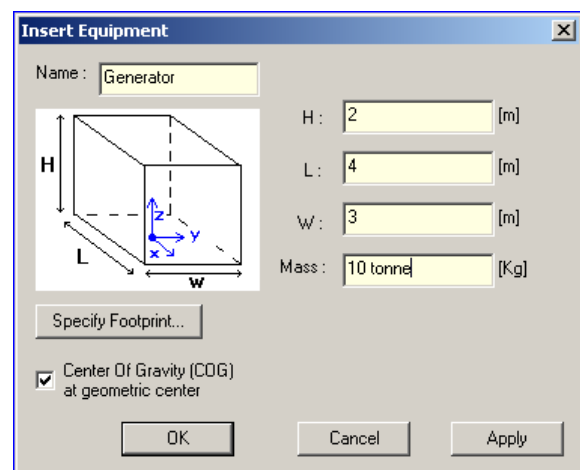
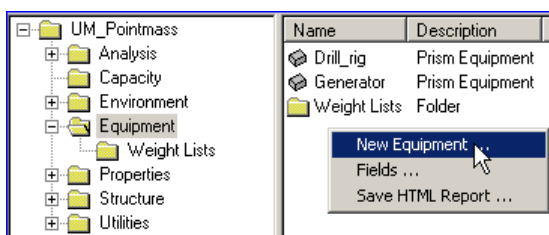
Equipments contain information about mass, local centre of gravity, size and footprint. It is also possible to use more high-level equipments (with no footprint information) imported from weight lists (from other systems or manually edited). Please refer to Volume I of the User Manual for a description on how to use weight lists.

Equipments may be created for each model. You may also build your library of equipments in a journal file or export to a XML-file. This will allow you to easily generate equipments you need for other projects.

4.5.1 Create equipments

The command **Insert/Equipment/Prism shape** defines the equipment with its mass, size, centre of gravity, and the footprint (or load transfer area). The example below shows the equipment *Generator* with its size and mass. The COG is calculated from the default rule which is in the middle of the box. Similarly, the default foot-print is the same as the bottom area of the box.

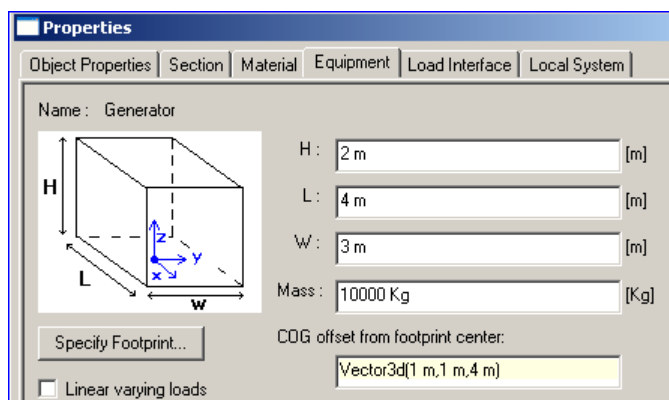
Equipments may also be defined and modified from the browser area.



4.5.2 Editing the COG and the footprint

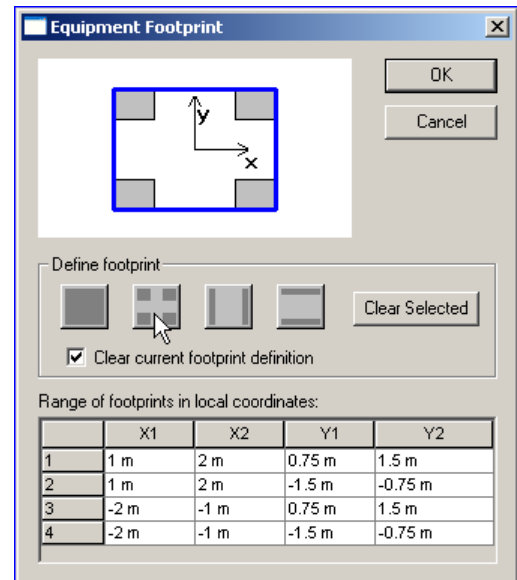
The COG is per default in the middle of the equipment. To change the COG select the equipment, **RMB**, *Properties*, and type in the new position of the COG. Note that this position is relative to the local coordinate system (origin in the middle of the bottom plane and local z-axis upwards). Note that the COG may be outside the equipment box. This origin at the bottom acts as a snap point when equipment is placed on structure.

In this case the COG is moved towards a corner and upwards.

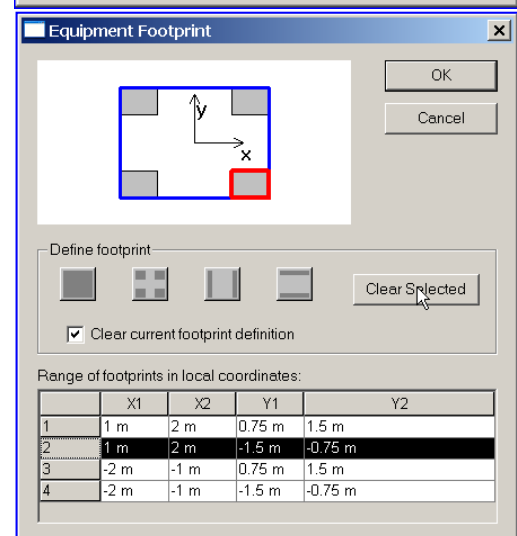


The footprint (or load distribution area) is per default the same as the bottom area defined by the length and the width of the equipment. To change the footprint, select the equipment, **RMB** and click *Specify Footprint*. There are 4 predefined footprint layouts, all of these may be edited and changed to meet the requirements. The example below shows how to change a 4 corner footprint may be changed to a 3 point footprint. Note that the footprint must be defined by an area.

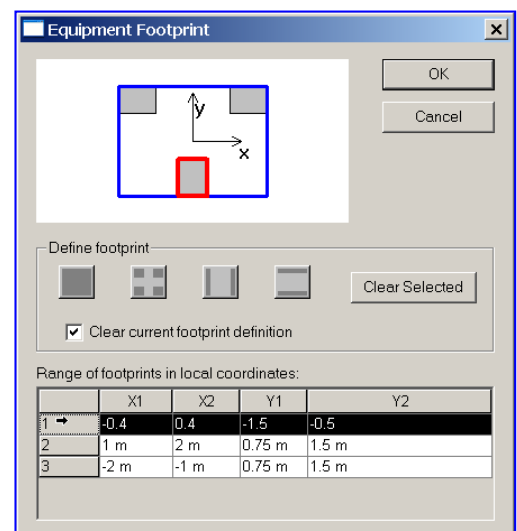
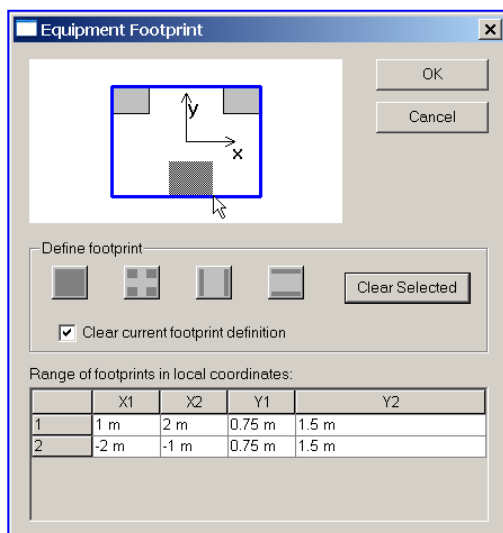
Change the footprint template from complete bottom to a 4 corner footprint.



Remove footprints 2 and 3 by selecting the footprints and click *Clear Selected*. You may also remove one of the footprints and edit the coordinate values to define the new position. Notice that all coordinates are local to the footprint.



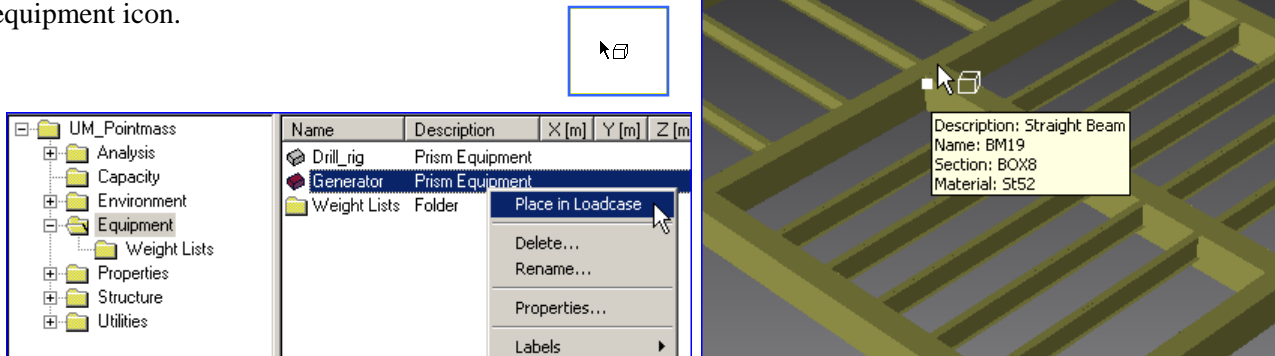
Use a rubber-band to indicate the new footprint. This will give you an approximate position and size of the new footprint and it is necessary to edit the values. In this case the values are modified to $X(1) = -0.4\text{ m}$, $X(2) = 0.4\text{ m}$, $Y(1) = -1.5\text{ m}$ and $Y(2) = -0.5\text{ m}$.



4.5.3 Placing the equipment

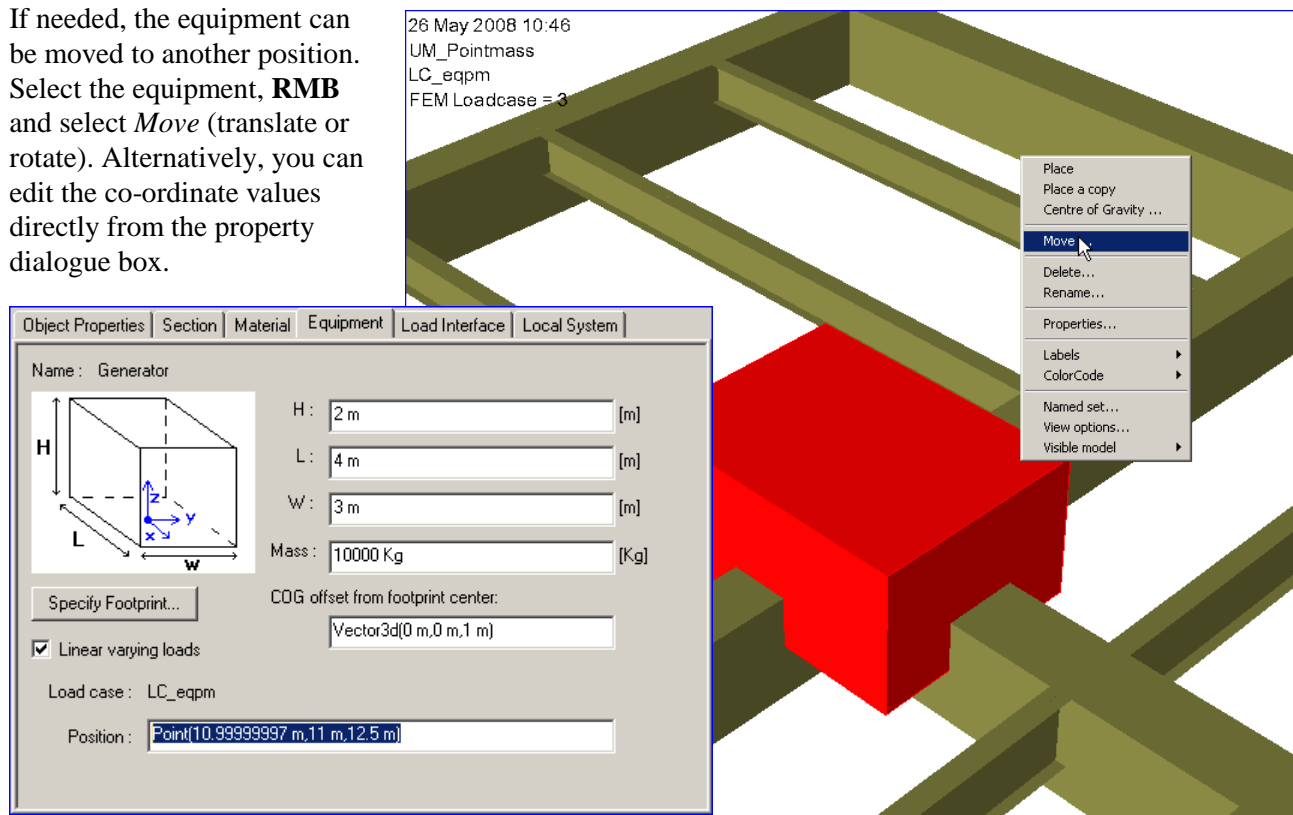
Prior to loading the structural model with equipments, a loadcase must be set to current (the active loadcase). The reason for this is that the equipment may be used in many loadcases at different positions. A loadcase is set to current by selecting it from the browser, **RMB** and *Set Current*. The example below shows how to place the equipment *Generator* to the deck structure in loadcase *LC_eqpm* and how to move it to another position.

Select the equipment from the browser, **RMB** and use *Place in Loadcase*. When you move the mouse to the graphical window, the mouse symbol includes an equipment icon.

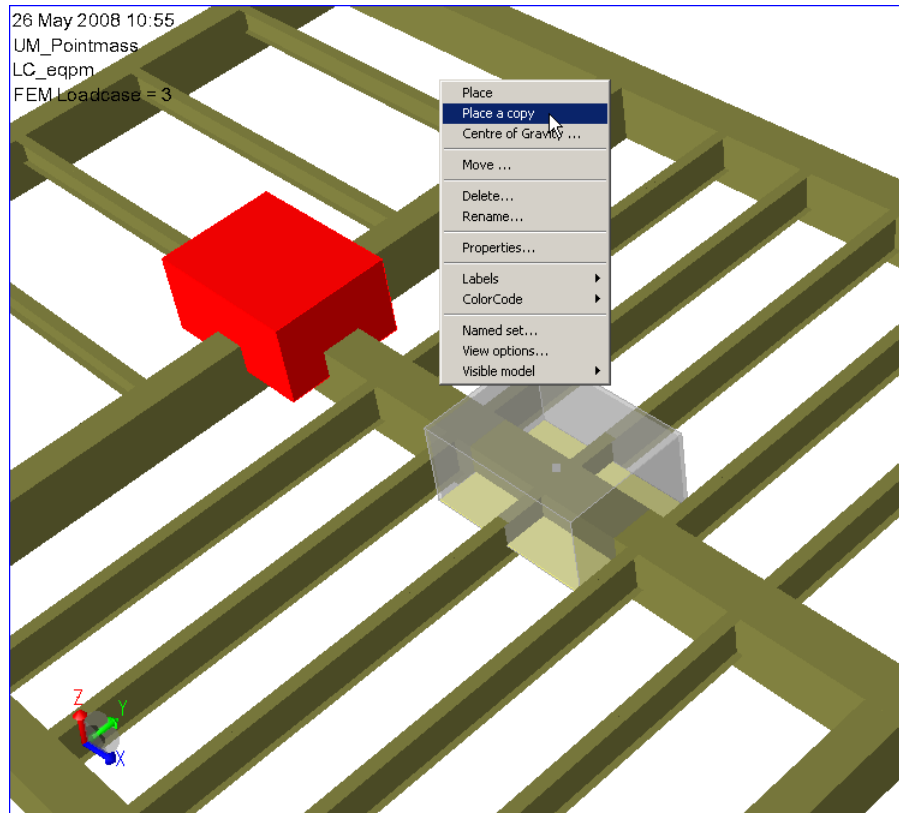


The equipment is now placed when you click it to a snap point. The equipment is connected to the snap point in the centre of the equipments bottom plan (the equipments local origin).

If needed, the equipment can be moved to another position. Select the equipment, **RMB** and select *Move* (translate or rotate). Alternatively, you can edit the co-ordinate values directly from the property dialogue box.



Equipments can be used once in a single loadcase. A copy of the equipment may be used in the same loadcase but at another position(s). Select the equipment, **RMB** and *Place a copy*. You place the copy in the same way as described above. This example shows that a new equipment is made and positioned next to the original equipment. The new equipment may be renamed from the default *Equipment**.



For more details on how to align equipments to sloped decks, at the side of a deck or below a deck see Volume I of the user manual.

4.5.4 Creating forces from placed equipments

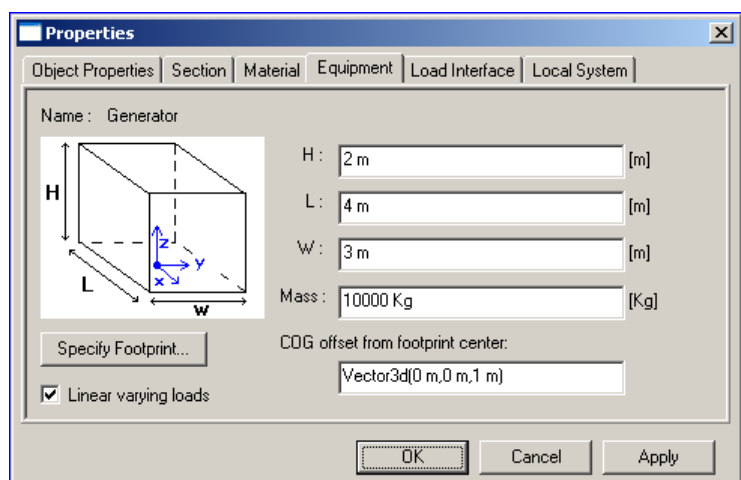
There are two different types of load calculation rules.

- Linearly varying loads. This method will always ensure equilibrium between the conceptual load (equipment mass x acceleration accounting for the COG) and the applied load (the line loads as generated by GeniE)
- Constant loads. This method will not ensure equilibrium.

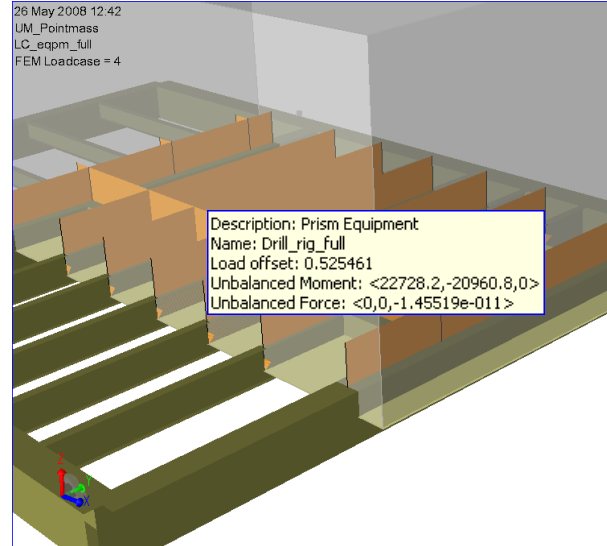
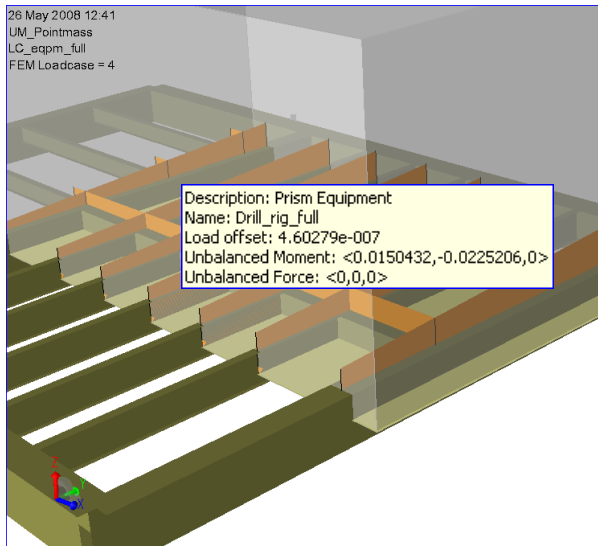
Default option is to calculate linearly varying loads. By de-activating the tick-off for linearly varying loads, the program calculates constant line loads. You may want to do this when e.g. working with uniform distributed loads (UDL).

The equipment loads are always calculated when creating a finite element mesh wither manually or as a result of running an analysis.

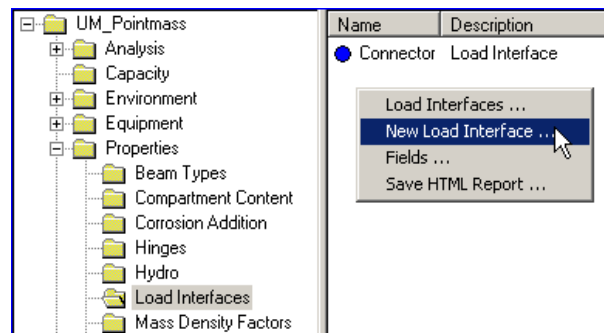
You may also force the program to calculate loads by selecting the loadcase from the browser, **RMB** and *Generate Equipment Loads*. You normally do this when you make up the loadcases and you want to verify the applied loads.



The two examples below show the difference between linear and constant – note that the mouse tooltip called *Load Offset* indicates the horizontal distance between the COG of the equipment mass and the COG for the applied load. The load offset is an indication of the un-balance introduced by choosing constant loads. By using linearly varying loads, the load offset is 4.60E-06 m (in other words equilibrium), while the constant line load option yields a load offset 0.53 m (a significant offset).



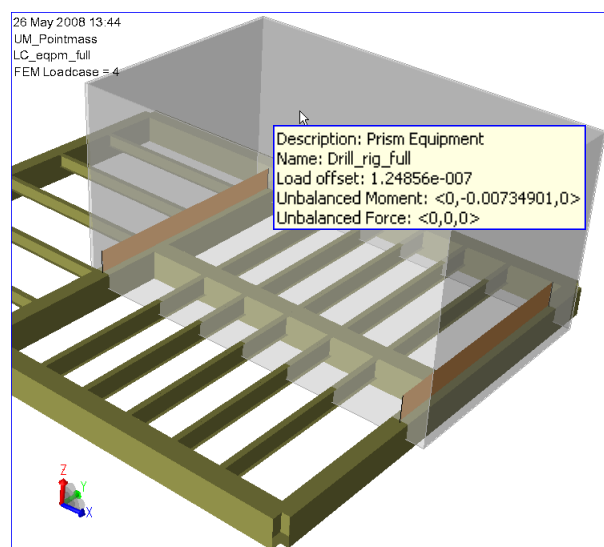
All beams that are intersecting the footprint will receive loads. In case you want some of the beams to receive loads you can connect the equipment with the beams in question by using a *Load Interface* that will overrule the global default. A load interface is defined from **Edit/Properties/Load Interface** or the browser. The same load interface can be used for various beams, equipments and loadcases.



In the example to the right the load interface *Connector* is applied to two beams and the equipment. Notice that it is necessary to apply the load interface to both beams and equipments. As can be seen, there are no loads generated for the intermediate beams.

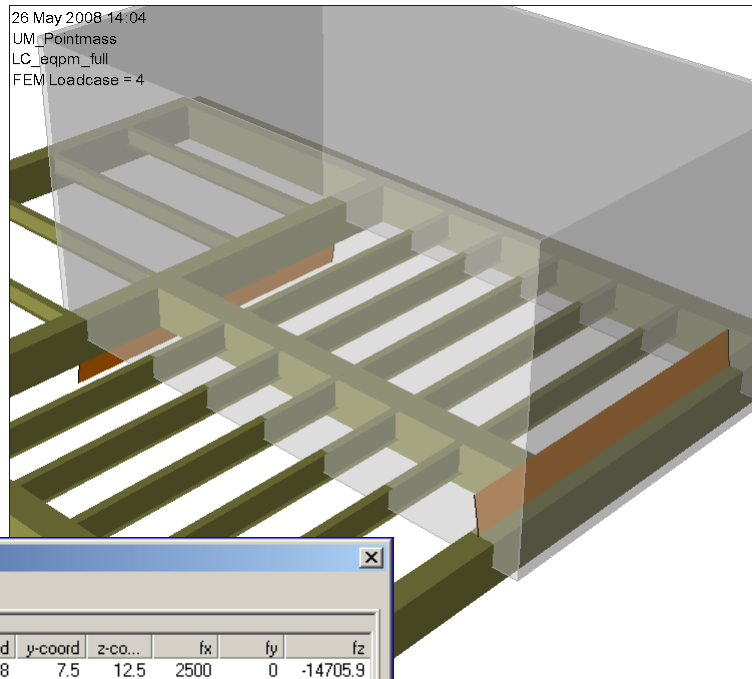
A relevant scenario for using load interfaces is when you want to load the primary stiffeners, but not the secondary stiffeners.

The footprint has been removed for better visibility.



The above loads assume a constant vertical acceleration field (gravity is one example of such). Adding a horizontal acceleration (10 m/s^2) will introduce shear forces and a force couple. This is shown to the right where the acceleration field is horizontal only. Note that when placing equipments along a vertical or sloped wall, both vertical forces (shear force) and horizontal forces (the force couple) are created.

The beam forces in this case are shown below (see following Sections on how to document the forces):



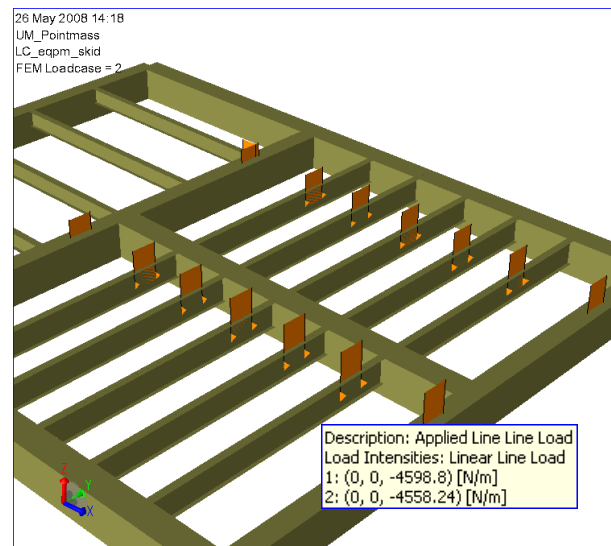
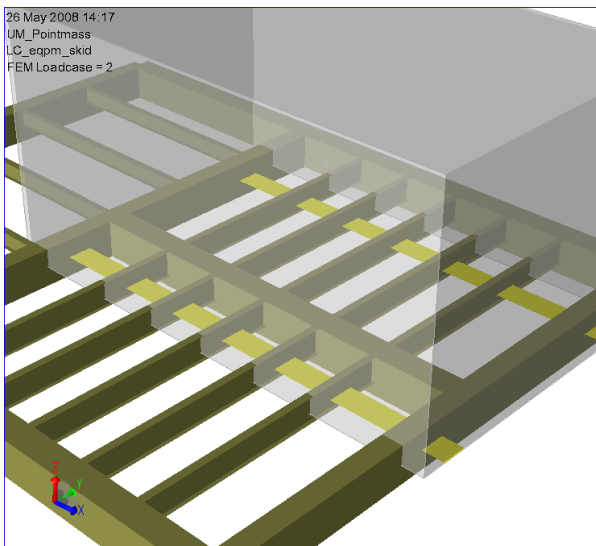
Load Case Properties: LC_eqpm_full

General | Equipment | Loads | Rotation Field | Design Condition

Load Generator	Structure	Description	x-coord	y-coord	z-co...	fx	fy	fz
Drill_rig_full	BM56	Applied Line Line Load.pos1	28	7.5	12.5	2500	0	-14705.9
Drill_rig_full	BM56	Applied Line Line Load.pos2	28	19.5	12.5	2500	0	-14705.9
Drill_rig_full	BM58	Applied Line Line Load.pos1	11	7.5	12.5	2500	0	14705.9
Drill_rig_full	BM58	Applied Line Line Load.pos2	11	19.5	12.5	2500	0	14705.9

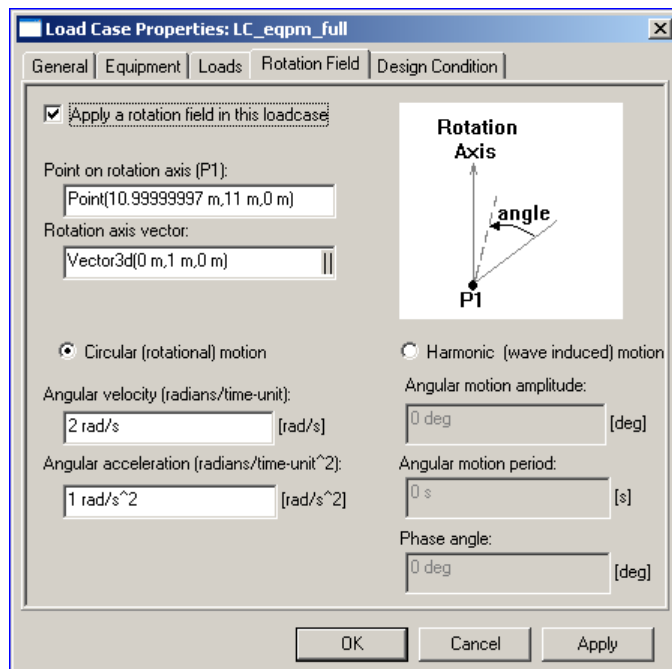
OK Cancel Apply

The example below shows the loads generated when using a footprint different than the bottom plan of the equipment. The footprint may be of any type as long as it is an area. In this case two skid-beams are simulated (yellow colour below). As can be seen, only the parts of the beams intersecting with the footprint receive loads. The equipment has been removed to easier see the loads. There are no load interfaces in this example.

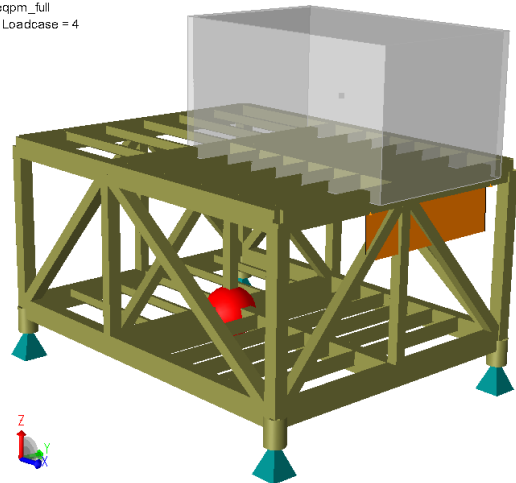


The previous examples of applied loads have all assumed a constant acceleration field. The loads may be generated based on a circular (rotational) motion or a harmonic (wave induced motion).

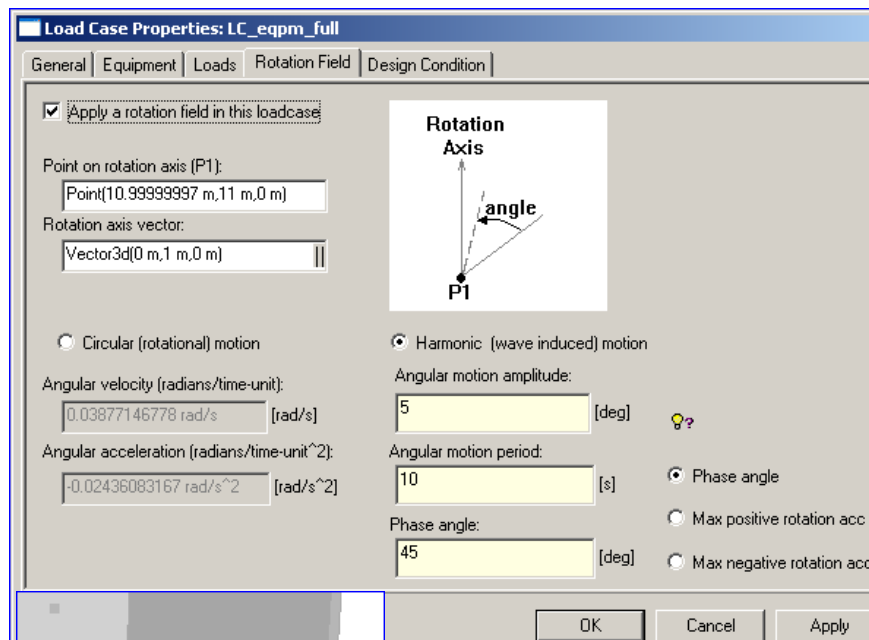
The rotation point is highlighted to the right. By using a rotation axis vector in global y-direction and angular velocity and accelerations this will set up a line load as shown. In this case the constant acceleration is set to zero.



26 May 2008 15:01
UM_Pointmass
LC_eqpm_full
FEM Loadcase = 4

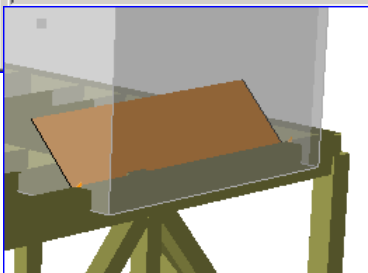
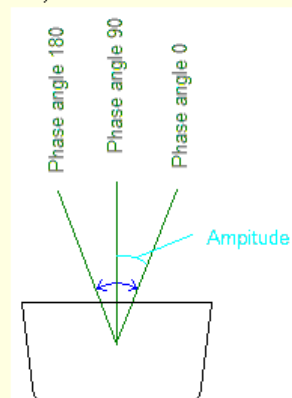


In the example below a harmonic motion is used.



Phase Angle

- The motion angle is at the maximum when the phase angle is 0.
- When gravitational acceleration is not included the maximum acceleration will usually occur at 0 and 180 deg phase angle for small rotational amplitudes.
- When gravity acceleration is included the maximum total acceleration field will usually occur at 90 or 270 deg phase angle for points below the rotation center (where gravity and rotational acceleration have the same direction).



4.5.5 Verify the applied loads

There are several ways of verifying that the equipments are placed and loads calculated as intended.

You may graphically verify by moving the mouse over the equipment where the mouse tooltip will show you some of the details primarily focusing the quality of the computations. For a balanced system, the load offset and unbalanced force and moments should be small.

Description: Prism Equipment
Name: Drill_rig_full
Load offset: 1.24856e-007
Unbalanced Moment: <0,-0.00734901,0>
Unbalanced Force: <0,0,0>

More detailed information may be found from the loadcase property dialogue.

The front property page summarises the mass of all equipments added to the loadcase. This leads to the *conceptual load* (masses x accelerations). A balanced system should have the same *conceptual load* as the *applied load* (sum of all the computed beam forces).

Load Case Properties: LC_eqpm_skid

General | Equipment | Loads | Rotation Field | Design Condition

Environment
Acceleration field: $\text{ctor3d}(0 \text{ m/s}^2, 2.0 \text{ m/s}^2, -9.80665 \text{ m/s}^2)$

Structural Analysis Load and Mass management
☒ Delete Explicit Loads
☐ Generate Applied Loads
☒ Represent Equipment as loads
☐ Represent Equipment as loadcase-independent mass:
☐ Include structure self-weight in structural analysis ☒ Include structure mass with rotation field

Sum over Equipments
 Mass [Kg]: 6000
 COG [m]: (19.75, 13.5, 17.5)
 Applied load [N]: $F_x=0, F_y=0, F_z=-58839.9$
 Conceptual load [N]: $F_x=0, F_y=0, F_z=-58839.9$

Sum
 Explicit conceptual load [N]: No loads
 Total applied load [N]: $F_x=0, F_y=0, F_z=-58839.9$

☒ FEM Loadcase number: 2
☒ Display in Input Units
☐ Display in Database Units

OK Cancel Apply

The tab *Equipment* lists the equipments that are part of the loadcase and where they are placed. Notice that the co-ordinate values are relative to the local origin (the snap point) of the equipments.

Load Case Properties: LC_eqpm_skid

General | Equipment | Loads | Rotation Field | Design Condition

Equipment	Description	Mass	X	Y	Z
Drill_rig_skid	Prism Equipment	6000	19.75	13.5	12.5

OK Cancel Apply

Detailed information about the line loads may be found from the tab *Loads*. Remember to activate the radio button "Applied Loads" to see the loads.

Load Case Properties: LC_eqpm_skid

General | Equipment | Loads | Rotation Field | Design Condition

Load Generator	Structure	Description	x-coord	y-coord	z-coord	fx	fy	fz
Drill_rig_skid	BM32	Applied Line Line Load.pos1	24.65	8.5	12.5	0	0	-4598.8
Drill_rig_skid	BM32	Applied Line Line Load.pos2	24.65	9.5	12.5	0	0	-4558.24
Drill_rig_skid	BM32	Applied Line Line Load.pos1	24.65	17.5	12.5	0	0	-4233.69
Drill_rig_skid	BM32	Applied Line Line Load.pos2	24.65	18.5	12.5	0	0	-4193.12
Drill_rig_skid	BM56	Applied Line Line Load.pos1	28	8.5	12.5	0	0	-4840.54
Drill_rig_skid	BM56	Applied Line Line Load.pos2	28	9.5	12.5	0	0	-4799.97
Drill_rig_skid	BM56	Applied Line Line Load.pos1	28	17.5	12.5	0	0	-4475.43
Drill_rig_skid	BM56	Applied Line Line Load.pos2	28	18.5	12.5	0	0	-4434.86
Drill_rig_skid	BM58	Applied Line Line Load.pos1	11	8.5	12.5	0	0	-3613.81
Drill_rig_skid	BM58	Applied Line Line Load.pos2	11	9.5	12.5	0	0	-3573.24
Drill_rig_skid	BM58	Applied Line Line Load.pos1	11	17.5	12.5	0	0	-3248.7
Drill_rig_skid	BM58	Applied Line Line Load.pos2	11	18.5	12.5	0	0	-3208.13
Drill_rig_skid	BM27	Applied Line Line Load.pos1	19.65	8.5	12.5	0	0	-4238
Drill_rig_skid	BM27	Applied Line Line Load.pos2	19.65	9.5	12.5	0	0	-4197.43
Drill_rig_skid	BM27	Applied Line Line Load.pos1	19.65	17.5	12.5	0	0	-3872.89
Drill_rig_skid	BM27	Applied Line Line Load.pos2	19.65	18.5	12.5	0	0	-3832.32

☐ Explicit Loads ☒ Applied Loads ☒ From Equipments ☐ From Explicit Loads ☐ Display Unit Notations

OK Cancel Apply

Load Generator	Structure	Description	x-coord	y-coord	z-coord	fx	fy	fz
Drill_rig_skid	BM32	Applied Line Line Load.pos1	24.65	8.5	12.5	0	0	-4598.8
Drill_rig_skid	BM32	Applied Line Line Load.pos2	24.65	9.5	12.5	0	0	-4558.24
Drill_rig_skid	BM32	Applied Line Line Load.pos1	24.65	17.5	12.5	0	0	-4233.69
Drill_rig_skid	BM32	Applied Line Line Load.pos2	24.65	18.5	12.5	0	0	-4193.12

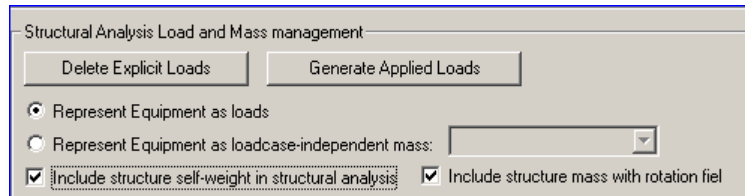
You can also document the equipments and the loads from the *File/Save report*.

4.6 Masses and inertia loads

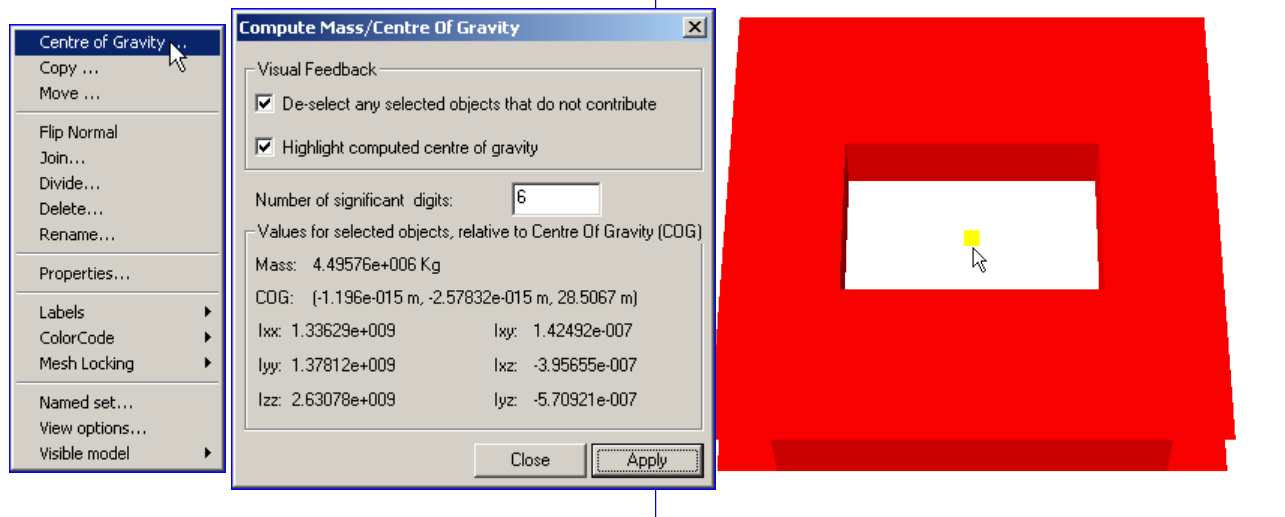
The three sources of masses - structural mass, point mass and equipment mass – that can contribute to the inertia loads. Structural mass and point mass is the same for all loadcases if added to a loadcase.

4.6.1 Structural mass

The structural mass is a result of material density and volume of the structure. GeniE will compute the mass and the centre of gravity. To include the mass in a loadcase (mass x gravity) of a finite element model you need to activate this in the load-case property.

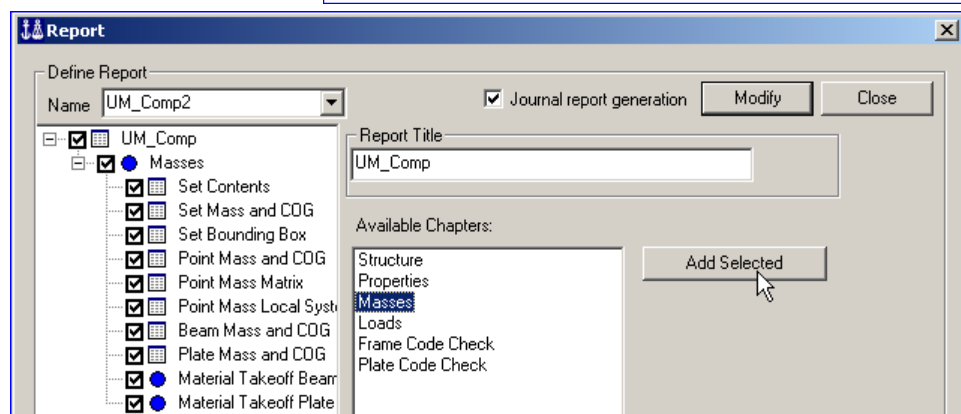


You can check the mass of the entire structure or parts of it from the graphical view or from a report. In the graphical view you select the object(s), **RMB** and *Centre of Gravity*. The mass, centre of gravity and inertia moments are calculated.



To find the mass and centre of gravity from the report you select **File/Save Report** and add the necessary chapters for masses in the report.

In this case the named set "UM_set".



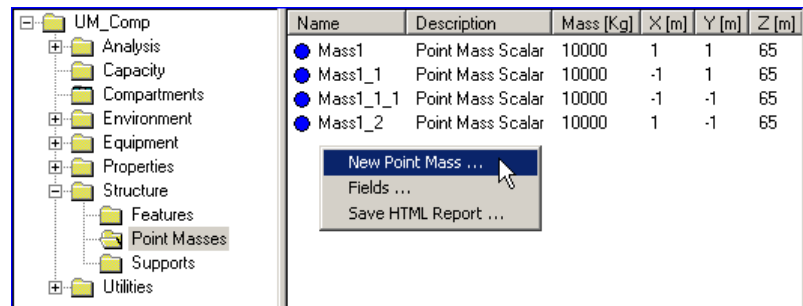
Set	About	Mass [Kg]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	IOX [Kg*m^2]	IYY [Kg*m^2]	IZZ [Kg*m^2]
Total	Origo	2.12679e+007	1.7516e-015	2.33661e-016	10.1314	2.08586e+010	1.8813e+010	2.61051e+010
Derrick	Origo	21473	-2.11776e-017	0	48.0207	5.18984e+007	5.28953e+007	2.7973e+006
Morison	Origo	164752	0	0	7	3.20843e+007	1.31401e+008	1.4734e+008
UM_set	Origo	4.49576e+006	-1.196e-015	-2.57184e-015	28.5067	4.98969e+009	5.03153e+009	2.63078e+009

- Scale the mass, and update the materials both in GeniE and on the FEM-file. By this you can also verify the new material names and connectivities in GeniE. You need to use the command **Tools/Properties/Create Scaled Materials** to do this.

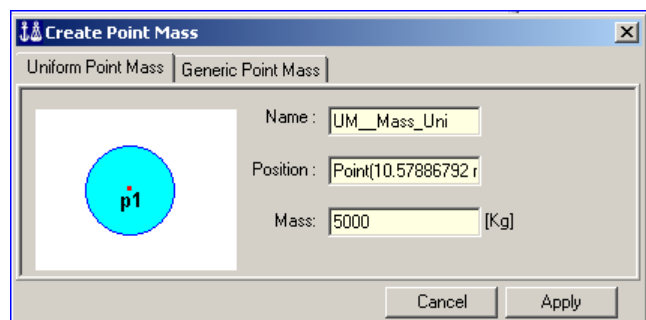
4.6.2 Point mass

Point masses may be added in addition to structural mass. The point masses contribute to a loadcase in the same way as structural mass; i.e. each loadcase must have the option “Include structure self-weight in structural analysis”. For a dynamic or eigen-value analysis both the structural mass and point masses are part of the mass matrix per default.

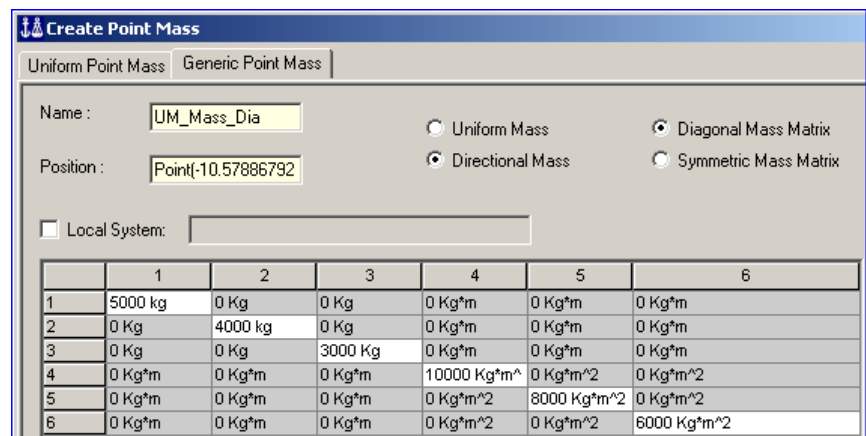
Point masses are inserted from **Insert/Mass** or from browser as shown.



There are two options for point masses. The **Uniform Point Mass** has same mass contribution in global x, y and z directions.



The other option is to define a **Generic Point Mass**. There are several options and in this case a directional and diagonal mass matrix has been defined.



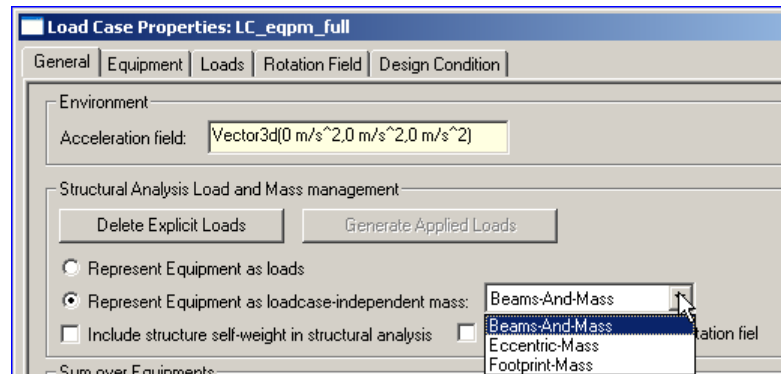
The point masses are documented in a report as well as from the graphical view.

Name	Description	Mass [Kg]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	IOX [Kg*m^2]	IOY [Kg*m^2]	IOZ [Kg*m^2]	IYY [Kg*m^2]	IYZ [Kg*m^2]	IZZ [Kg*m^2]
UM_Mass_Dia	Point Mass Matrix	3000	-10.5789	7.09434	46	10000	0	0	8000	0	6000
UM_Mass_Uni	Point Mass Scalar	5000	10.5789	7.09434	46						

4.6.3 Equipment mass

The mass from equipments can be added to the mass matrix by specifying “Represent Equipment as loadcase-independent mass”. There are a number of options for this alternative. Each one is described in the following.

Notice that each loadcase assigned the equipment mass alternative will add to the total mass matrix.

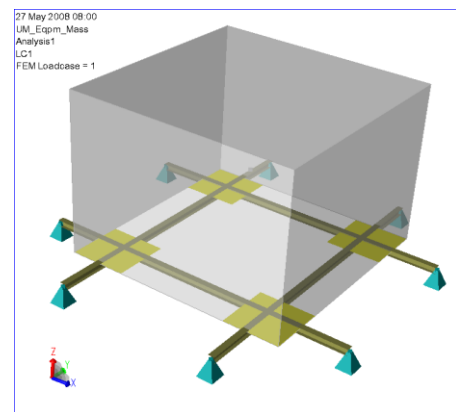


There are three options for creating mass models (from basic loadcases only):

- Eccentric mass – should be used for hydrodynamic dynamic analysis only
- Beams and Mass – for hydrodynamic dynamic analysis followed by structural analysis, or dynamic structural analysis
- Footprint mass – all masses are flushed down to the footprint level neglecting the equipment COG

To illustrate the differences between the two first alternatives, the following model is used for references. The model consists of four beams, one equipment with four footprints, and boundary conditions (free to rotate in all degrees of freedom).

For more details, see Volume I of the User Manual.



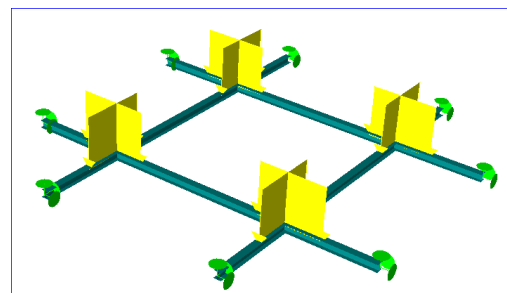
4.6.3.1 Mass model for hydrodynamics

A common scenario when making a mass model for hydrodynamics is when the GeniE model contributes to the overall mass model of a floater. The complete mass model is now (together with a panel model) analysed in HydroD to find e.g. rigid body motions and global sectional loads.

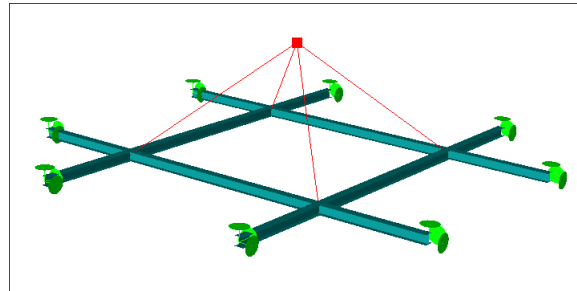
When sectional loads are to be computed by HydroD, a description of the mass *distribution* is required. The technique of modelling equipments rather than explicit loads may lead to a faster definition of the mass model.

In the example below a mass model using the eccentric mass option has been used. Finite element of type GMAS (one node mass element) are inserted. The eccentric mass elements have their mass centre always at the same position as the equipment local cog. Whenever there is an interface between finite element nodes and footprints, eccentric one node mass elements are created (sum of masses of mass elements = equipment mass).

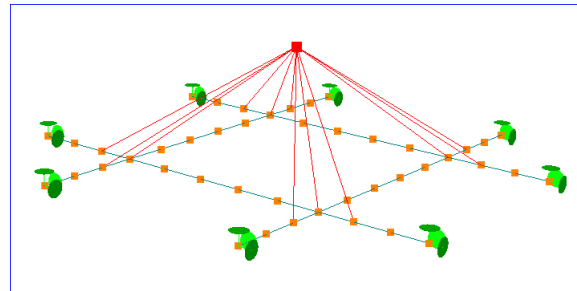
Line loads created when equipments are represented as loads. See previous sections for details.



Four eccentric mass elements created when equipment adds to mass model. There are four finite element nodes inside the footprints.

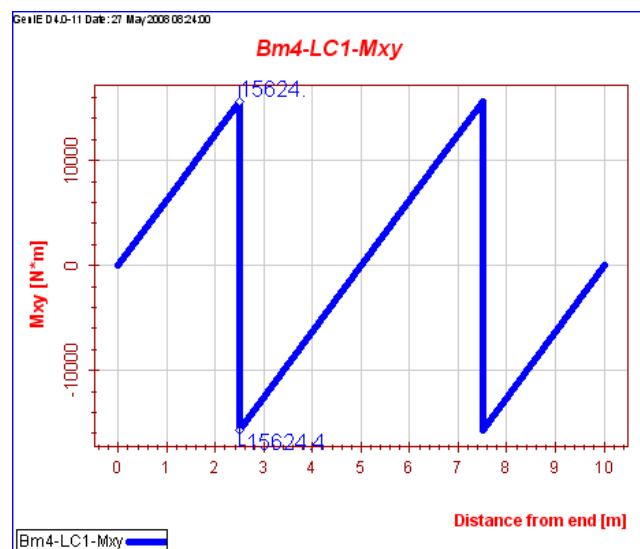
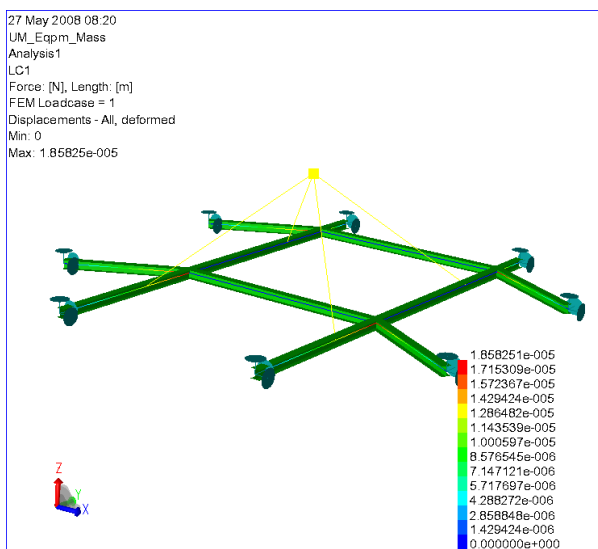


Mesh density increased so that there are twelve finite element nodes inside the footprints. Hence, twelve eccentric mass elements are created.



When applying a horizontal acceleration to a loadcase containing the equipment, correct displacements are computed, but the bending moments are not correct (the peaks are much higher) compared to a real case. The reason is that the connection between the one node eccentric mass elements and structure is fixed in all degrees of freedom and hence moments are computed. In a realistic case, there are only vertical and lateral forces to be transferred from the equipment and no moment transfer (these forces will set up some moment effects, but not as large as shown below). This is the reason why this approach should only be used for hydrodynamic analysis; if dynamic structural analysis shall be carried out the next alternative for mass representation should be used.

The pictures below show displacements and the undesired bending moments due to a horizontal acceleration component. The displacements will be realistic, but the moments are unrealistic.



The option "Eccentric-Mass" should be used for a subsequent hydrodynamic analysis only.

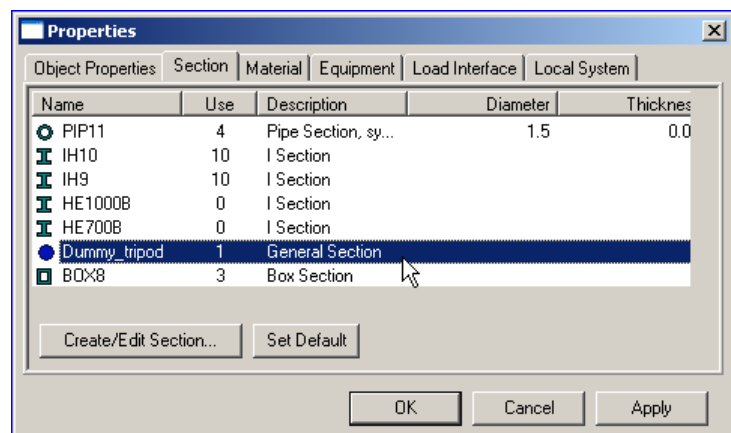
4.6.3.2 Mass model for structural dynamics

When creating a mass model for structural dynamics it is important to avoid the undesired bending moments as discussed above. GeniE will do this automatically by inserting additional elements with hinges (i.e. no rotational connections) between the mass element and the structure. This technique is referred to as tripod, tent, or “chicken feet”. To be able to do so, it is required to add sectional and material data to the equipment. This information is added from the equipment property sheet.

Section type *Dummy_tripod* and an equivalent material type is assigned to the equipment from the equipment property sheet (select the equipment, **RMB**, *Properties*).

Normally, equivalent sections are used addressing no side effects when e.g. performing eigenvalue analysis.

Also, it is common to use a material property with close to zero density (the equipment mass is represented from a point mass and not the structure associated with the equipment).

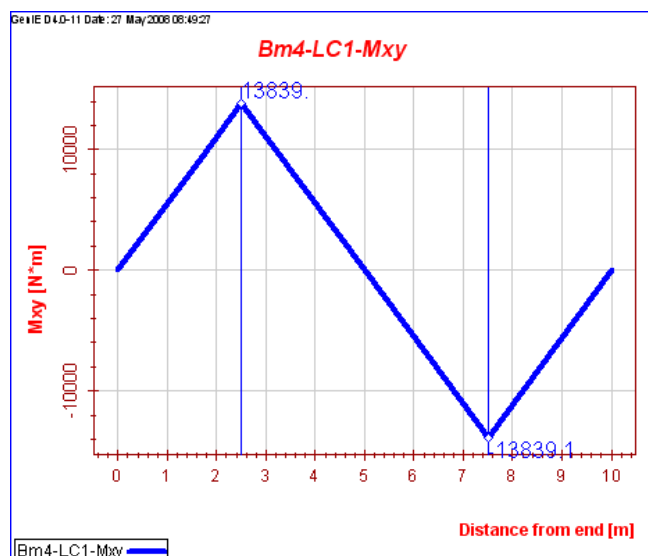
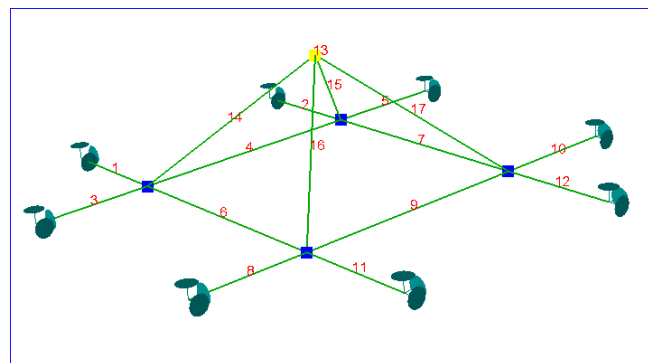


In the following example the option *Beams and Mass* for representing equipments as loadcase independent mass has been used.

A finite element model has been created. The mass of the equipment is represented as one point mass (in this case finite element number 13) which is connected to the structure with beam finite elements 14, 15, 16, and 17.

Hinges are inserted where connected to the structure (free to rotate around local y and z axis) to avoid the undesired bending moments as in the case by representing equipments with one node eccentric mass elements. Hinges are inserted at the lower ends of finite element 14, 15, 16, and 17 (select these elements, **RMB**, *Labels and Hinge symbols* to see the blue labels indicating hinges).

A better and more realistic moment distribution is now achieved. The peak of the bending moment has been reduced from 15624 Nm to 13839 Nm (or 13%) in this case. The differences in result depend highly on structure, equipment properties, and location of equipments



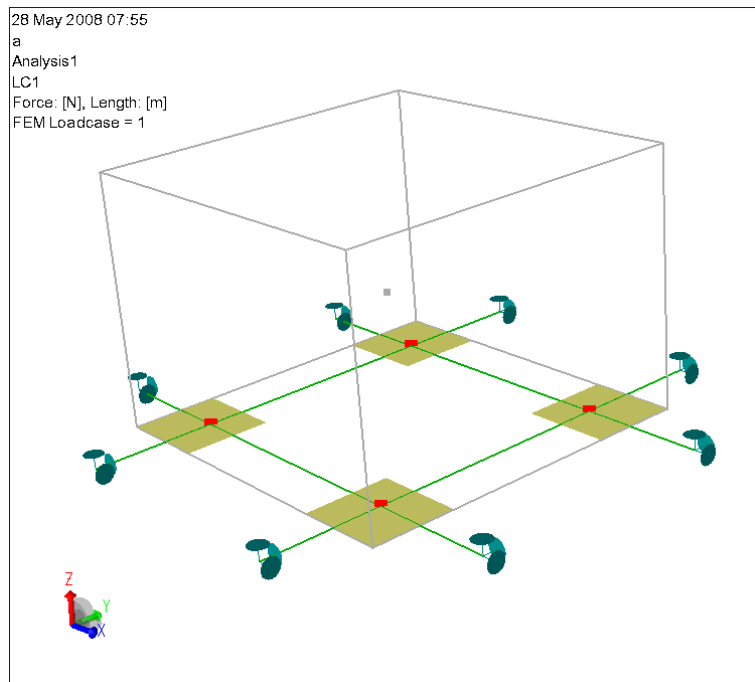
4.6.3.3 Mass model when neglecting eccentricities

It is possible to create a mass model neglecting the eccentricities of the equipments centre of gravity. This may be the desired mass model when working with large equipments to form uniform blanket loads (UDL) or blanket loads.

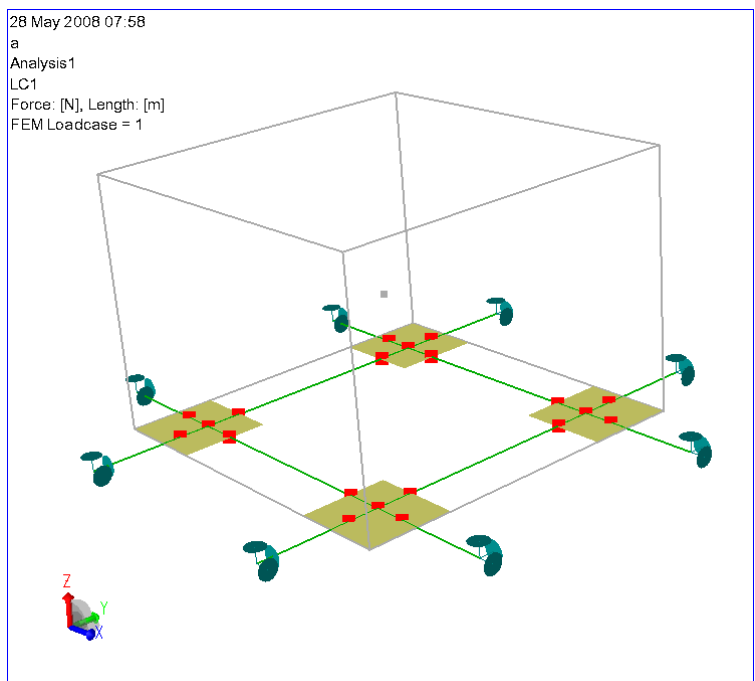
Beams-And-Mass
Eccentric-Mass
Footprint-Mass

For this alternative GeniE will calculate the mass elements as for the first option (*Eccentric-Mass*), but neglect all eccentricities. Hence, it is not necessary to associate section and material properties to the equipment in this case.

The mass elements are flushed down to the footprint level. There is one finite element node inside each of the footprint area.



In this case the mesh density has been refined so that there are five finite element nodes inside each footprint area. Hence, five mass elements are created per footprint area.



4.7 Verify and document loads and masses

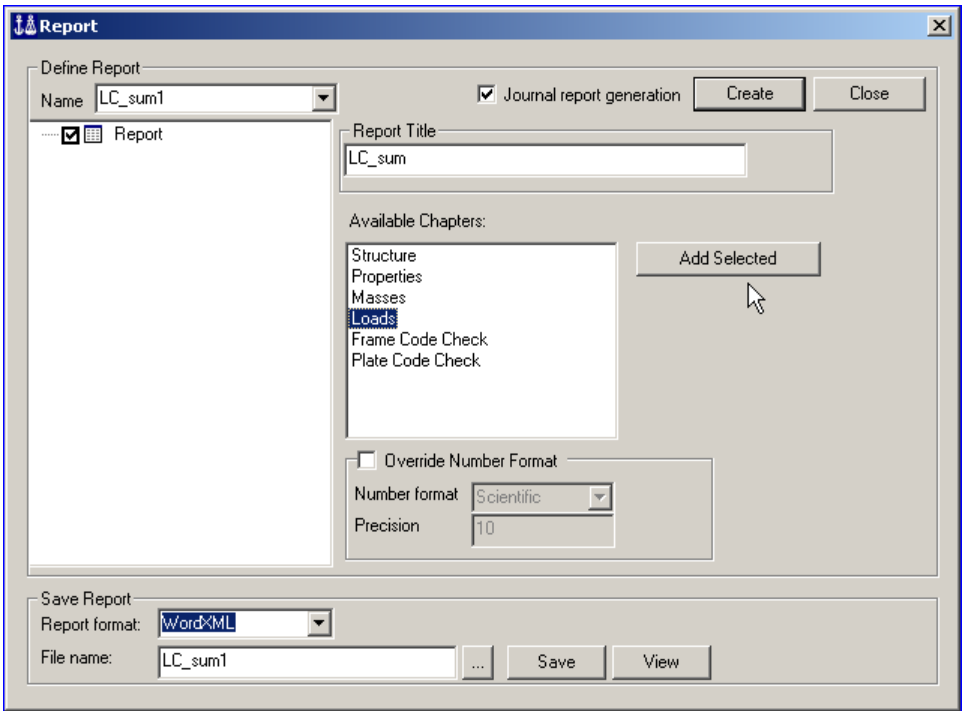
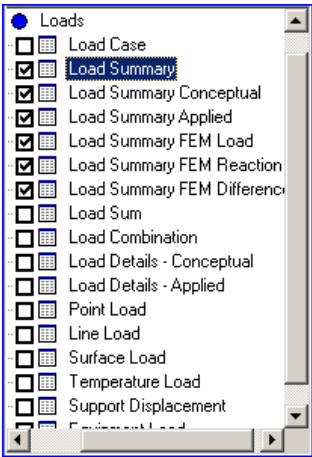
There are a number of ways to verify and document the loads, load combinations and masses. For detailed assessment of each load this is described in the previous Sections. For a more general overview you can use the print features in the *File/Save Report* and view it in your notepad editor, Internet Explorer, MS Excel or MS Word (notice you need MS Office 2003 edition or later).

The report generated from *File/Save Report* also includes the finite element loads used by the analysis program as well as the reaction forces from the analysis itself.


The example below shows the loads for a model with 4 basic loadcases and 1 load combination. The report is viewed in MS Word.

Select *Loads* and click *Add Selected* to include load reporting into the report.

The next step is to include the relevant parts to document.



This generates a report with the following content:

	Report: LC_sum1	Model Id: LC_sum1 Description: LC_sum Model file name: C:\Program Files\DNV\GeniE_D4010\Workspaces\LC_sum	Sign: nek Date: 07-May-2008 Last saved: 07-May-2008 14:40:34
Table of Contents			
1 Loads3 1.1 All loadcases3 1.1.1 All loadcases : Load Summary3 1.1.2 All loadcases : Load Summary Conceptual4 1.1.3 All loadcases : Load Summary Applied5 1.1.4 All loadcases : Load Summary FEM Load6 1.1.5 All loadcases : Load Summary FEM Reaction7 1.1.6 All loadcases : Load Summary FEM Difference8			

Some examples of the content – the *Load summary*:

LoadCase	FEM LC	LoadType	FX [N]	FY [N]	FZ [N]	MX [N*m]	MY [N*m]	MZ [N*m]	Count
LC1	1	Total Conceptual	0	0	-6536.44	-71900.9	93401.8	0	61
...	1	Total Applied	0	0	-6536.44	-71900.9	93401.8	0	61
...	1	Total FEM Load	5.68434e-014	0	-6536.44	-71900.9	93401.8	-6.25278e-013	0
...	1	Total FEM Reaction	-2.16005e-012	-2.27374e-013	6536.44	71900.9	-93401.8	2.54659e-011	0
...	1	Total FEM Difference	-2.10321e-012	-2.27374e-013	0	0	0	2.48406e-011	0
LC2	2	Total Conceptual	0	0	-58839.9	-794339	1.16209e+006	0	1
...	2	Total Applied	0	0	-58839.9	-794339	1.16209e+006	0	1
...	2	Total FEM Load	0	0	-58839.9	-794339	1.16209e+006	0	0
...	2	Total FEM Reaction	-8.68567e-011	-5.91172e-012	58839.9	794339	-1.16209e+006	1.03319e-009	0
...	2	Total FEM Difference	-8.68567e-011	-5.91172e-012	0	0	0	1.03319e-009	0
LC3	3	Total Conceptual	-8000	0	-60000	-810000	1.07e+006	108000	4
...	3	Total Applied	-8000	0	-60000	-810000	1.07e+006	108000	4
...	3	Total FEM Load	-8000	0	-60000	-809999	1.07e+006	108000	0
...	3	Total FEM Reaction	8000	-1.36424e-011	60000	809999	-1.07e+006	-108000	0
...	3	Total FEM Difference	0	-1.36424e-011	0	0	0	0	0
LC4	4	Total Conceptual	0	0	-10560	-116160	190382	0	10
...	4	Total Applied	0	0	-10560	-116160	190382	0	10
...	4	Total FEM Load	0	0	-10560	-116160	190383	0	0
...	4	Total FEM Reaction	-1.59162e-011	-1.02318e-012	10560	116160	-190383	1.34605e-010	0
...	4	Total FEM Difference	-1.59162e-011	-1.02318e-012	0	0	0	1.34605e-010	0
LoadComb1	5	Total Conceptual	-8000	0	-160980	-2.11201e+006	2.99777e+006	108000	76
...	5	Total Applied	-8000	0	-160980	-2.11201e+006	2.99777e+006	108000	76
...	5	Total FEM Load	-8000	0	-160980	-2.11201e+006	2.99777e+006	108000	0
...	5	Total FEM Reaction	8000	-2.32944e-011	160980	2.11201e+006	-2.99777e+006	-108000	0
...	5	Total FEM Difference	-1.42791e-010	-2.32944e-011	0	0	0	1.60071e-009	0

Some examples of the content – the *Load summary conceptual*:

LoadCase	FEM LC	LoadType	FX [N]	FY [N]	FZ [N]	MX [N*m]	MY [N*m]	MZ [N*m]	Count
LC1	1	Total Conceptual	0	0	-6536.44	-71900.9	93401.8	0	61
LC2	2	Total Conceptual	0	0	-58839.9	-794339	1.16209e+006	0	1
LC3	3	Total Conceptual	-8000	0	-60000	-810000	1.07e+006	108000	4
LC4	4	Total Conceptual	0	0	-10560	-116160	190382	0	10
LoadComb1	5	Total Conceptual	-8000	0	-160980	-2.11201e+006	2.99777e+006	108000	76

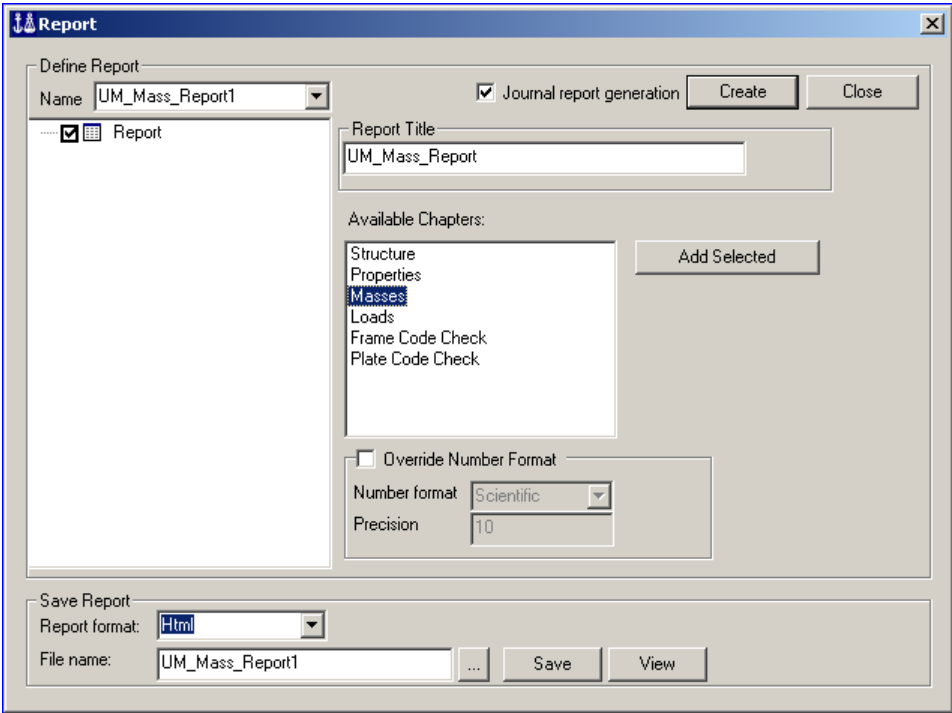
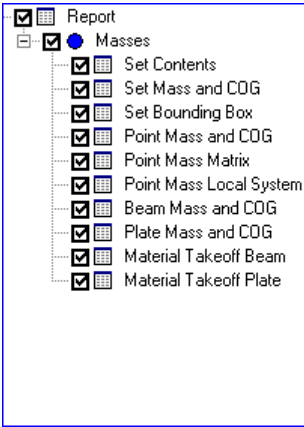
Some examples of the content – the *Load summary FEM difference* (difference between FEM forces and FEM reaction forces)

LoadCase	FEM LC	LoadType	FX [N]	FY [N]	FZ [N]	MX [N*m]	MY [N*m]	MZ [N*m]	Count
LC1	1	Total FEM Difference	-2.10321e-012	-2.27374e-013	0	0	0	2.48406e-011	0
LC2	2	Total FEM Difference	-8.68567e-011	-5.91172e-012	0	0	0	1.03319e-009	0
LC3	3	Total FEM Difference	0	-1.36424e-011	0	0	0	0	0
LC4	4	Total FEM Difference	-1.59162e-011	-1.02318e-012	0	0	0	1.34605e-010	0
LoadComb1	5	Total FEM Difference	-1.42791e-010	-2.32944e-011	0	0	0	1.60071e-009	0


To document the masses the procedure is the same as for loads (*File/Save Report*).

Select *Masses* and click *Add Selected* to include load reporting into the report.

The next step is to include the relevant parts to document.



The report contains a table of content as shown below.

 D4.0-12	Report: UM_Mass_Report1	Model Id: UM_Mass_Report1 Description: UM_Mass_Report Model file name: C:\Program Files\DNV\S\GeniE_D4012\Workspaces\UM_Mass_Report	Sign: nek Date: 28-May-2008 Last saved: 28-May-2008 08:09:58
Table of Contents			
1 Masses			3
1.1 Set Contents			3
1.2 Set Mass and COG			4
1.3 Set Bounding Box			4
1.4 Point Mass and COG			4
1.5 Point Mass Matrix			4
1.6 Point Mass Local System			4
1.7 Beam Mass and COG			4
1.8 Plate Mass and COG			4
1.9 Material Takeoff Beam			4
1.10 Material Takeoff Plate			4

Some of the sections are shown

The total mass as well as the mass per named sets including the COG can be added to the report.

Set	About	Mass [tonne]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	IOX [tonne*m^2]	IYY [tonne*m^2]	IZZ [tonne*m^2]
Total	Origo	2480.74	-0.191056	0.0252191	-54.506	1.27655e+007	1.27499e+007	580264
A1_Leg_Deck	Origo	2.84655	-6.00011	-6.00011	14.8138	739.132	739.132	204.959
A2_Leg_Deck	Origo	2.56338	6.00011	-6.00011	14.9721	678.099	678.099	184.57
B1_Leg_Deck	Origo	2.84655	-6.00011	6.00011	14.8138	739.132	739.132	204.959
B2_Leg_Deck	Origo	2.56338	6.00011	6.00011	14.9721	678.099	678.099	184.57
Boat_landings	Origo	12.9869	8.09198	-0.0371293	1.15022	255.992	900.513	1070.96
Cellar_deck_el13	Origo	50.7847	-2.24006	1.9629	12.7294	12592.9	11883.5	8018.11
Complete_Deck	Origo	287.406	-4.13284	-2.49085	18.1938	141087	131966	62602.9
Complete_Jacket	Origo	1086.57	-0.177848	0.00284107	-47.3484	3.36517e+006	3.36439e+006	224829
Conductors	Origo	288.406	3.0855	-1.61594e-010	-39.3837	703331	705174	4130.68
Deck_Row_1	Origo	19.9961	-6.00011	-0.547367	15.6708	6548.42	5764.15	2224.04
Deck_Row_2	Origo	18.9978	6.00011	1.40061	15.5444	5939.33	5394.28	1912.94
Deck_Row_3	Origo	17.1715	-14.0001	0.405199	15.9379	5969.91	7841.15	4860.1

The material take-off report lists both the relevant data for beams and plates. The volume, mass and surface area is calculated based on the eccentric length.

Material Name	Section Name	Centric Length [m]	Eccentric Length [m]	Volume [m^3]	Mass [tonne]	Density [tonne/m^3]	Section Circumference [m]	Surface area [m^2]
Mat_def	Cone_Middle	38.8191	38.8	0	0	0	0	0
Mat_def	HE200A	58.3901	58.3901	0.298081	2.33994	7.85	1.167	68.1412
Mat_def	HE200B	46.8502	46.8502	0.352782	2.76934	7.85	1.182	55.377
Mat_def	HE280A	57.2011	57.2011	0.52808	4.14543	7.85	1.644	94.0386
Mat_def	HE300B	343.768	343.768	4.90969	38.5411	7.85	1.778	611.219
Mat_def	HE400A	130.4	130.4	1.99148	15.6331	7.85	1.958	255.324
Mat_def	HE500A	138.003	138.003	2.63971	20.7217	7.85	2.156	297.533
Mat_def	HE600A	477.211	477.211	10.5082	82.4893	7.85	2.354	1123.36
Mat_def	IPE200	79.9704	79.9704	0.217903	1.71054	7.85	0.7888	63.0807
Mat_def	IPE270	54.3008	54.3008	0.238997	1.87613	7.85	1.0668	57.9281
Mat_def	P114_3x11_13	121.743	121.743	0.439181	3.44757	7.85	0.359084	43.716
Mat_def	P114_3x6	117.71	117.71	0.240294	1.88631	7.85	0.359084	42.2678

Material Name	Thickness Name	Area [m^2]	Volume [m^3]	Mass [tonne]	Density [tonne/m^3]
Mat_plates	Tck1	1277.64	10.2211	0.0102211	0.001
Mat_plates	Tck2	324.011	3.24011	0.00324011	0.001

In case you want to document the mass and COG per beam or plate the report will include the following.

Name	Description	Mass [tonne]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]
Bm1	Overlapping Straight Beam	22.5303	-14	-14	-81.1655
Bm2	Overlapping Straight Beam	22.5303	14	-14	-81.1655
Bm3	Overlapping Straight Beam	22.5303	-14	14	-81.1655
Bm4	Overlapping Straight Beam	22.5303	14	14	-81.1655
Bm5	Straight Beam	9.56071	-7.4319e-016	-14	-87
Bm6	Straight Beam	9.56071			
Bm7	Straight Beam	9.56071	7.4319e-016		
Bm8	Straight Beam	9.56071			

Name	Description	Mass [tonne]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]
PI1	Plate	0.00324011	-13	-2e-009	25.93
PI2	Plate	0.000768014	-10.0001	-2e-009	19.7
PI3	Plate	0.00233202	-4.25	11.5001	19
PI4	Plate	0.000576031	-3	-2e-009	19
PI5	Plate	0.000112474	3.00011	4.8285	19

5. APPLY BOUNDARY CONDITIONS

Boundary conditions are normally used to describe how a structure is fixed to its support conditions either it is on ground or other structural parts. Notice that there must be sufficient boundary conditions applied to the model to prevent rigid body motion in the structural analysis.

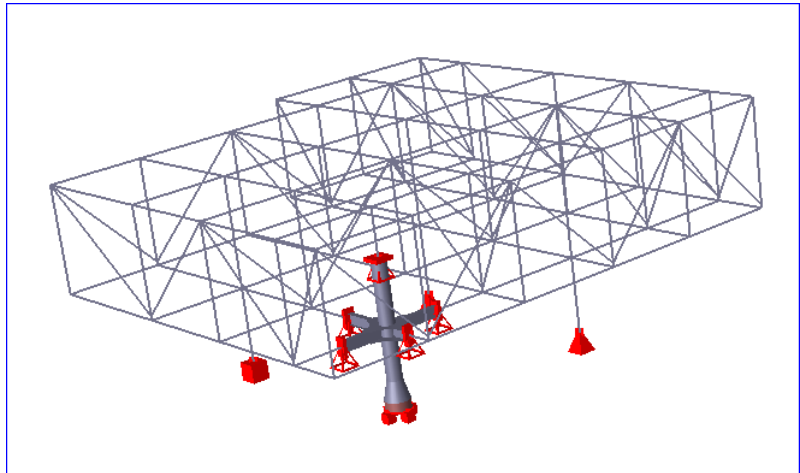
The boundary conditions are also used to describe additional conditions than how the structure is fixed:

- full fixation, i.e. fixed in all six degrees of freedom (this is the program default)
- free to translate or rotate in one or several degrees of freedom
- ground spring support defined either as a linear spring with stiffness properties in the translation degrees of freedom or as a full stiffness matrix
- prescribed displacements when the deformation of a certain point is known
- super-nodes used to define the connection nodes used in a super-element hierarchy
- rigid link support to define a governing point, the related points and the relations between (also known as master slave)

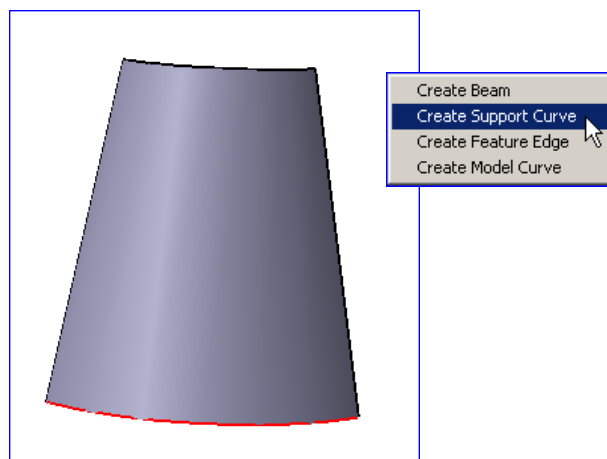
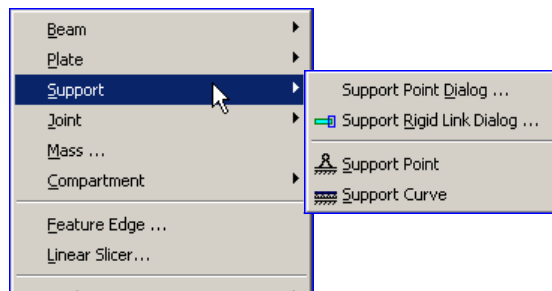
Boundary conditions may be relative to the global co-ordinate system or aligned with a transformed co-ordinate system. All alternatives are described in the following except for prescribed displacements which is documented in the previous Chapter.

The boundary conditions may be inserted at support points or along an edge (along a beam, along a plate, inside a plate). To explain the various boundary conditions a common model is used.

The beams are shown in wireframe mode and the various support points and lines are highlighted.



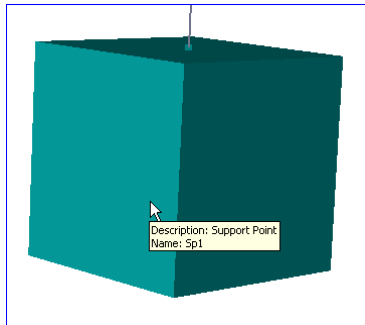
Support points or lines are inserted from **Insert/Support** or from a line or edge (double click a plate to see the edges).



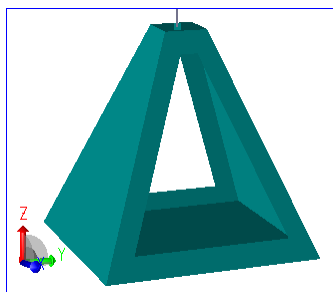
5.1 Fixation and rotation supports

A support point is inserted from *Insert/Support/Support Point Dialog* or *Support Point*. Alternatively you can also define a support point from the browser.

The support point dialogue is shown to the right. The input parameters are typically the position and type of boundary condition. This example shows the program default which is fixed in all six degrees of freedom. The graphic symbol is shown below.

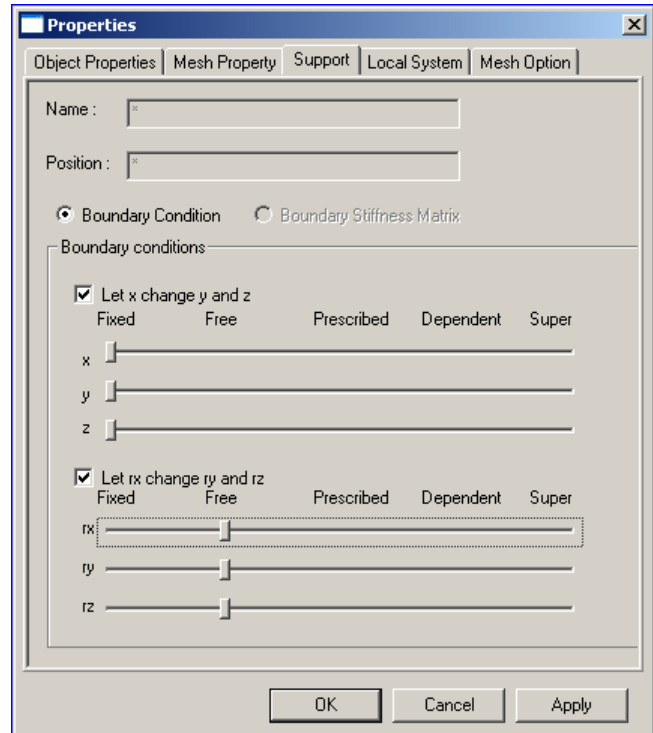
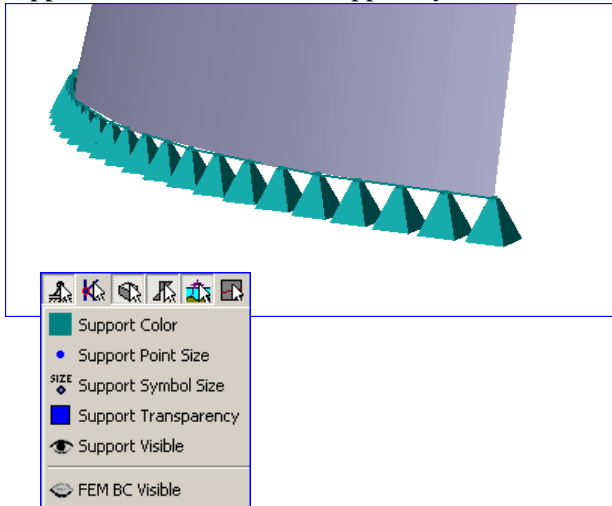


The boundary conditions may be modified during input mode or by editing an existing support condition (select the support, **RMB** and *Properties*). In this case it is fixed in global y and z directions. As can be seen, the graphic view changes.



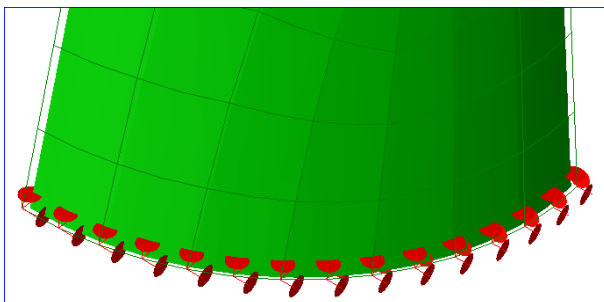
Support curves are inserted from **Insert/Support/Support Curves** or from a line or edge. In both cases a support curve is defined with boundary conditions according to the program default setting (fixed in all six degrees of freedom). To represent other boundary conditions edit the support curve by selecting the support curve, RMB and Properties. In the example below, the boundary conditions are modified to simulate a hinged connection, i.e. free to rotate in all directions.

The support curve shows the boundary conditions along the curve. The number and graphical size depends on the settings you specify from the support visualisation icon (support symbol size).

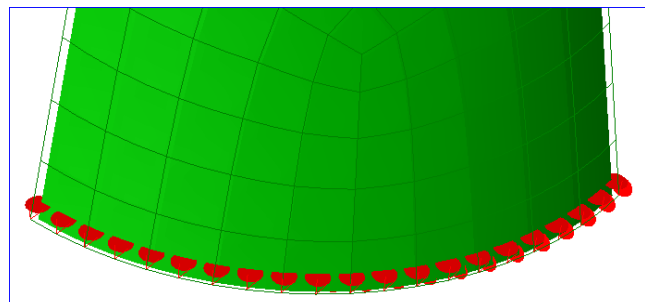


The support curve will ensure that boundary conditions are added to each finite element node along the curve. This means that you can easily change the mesh density without altering the boundary conditions.

Typical views of boundary conditions applied to a finite element model are shown below.



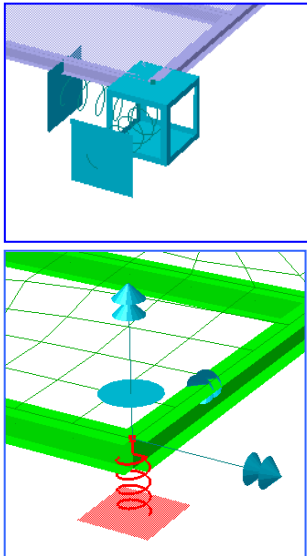
*Fixed in translation DOF, Free in rotation DOF
Mesh density 0.15m*



*Fixed in global y and z-directions, all other fixed
Mesh density 0.10m*

5.2 Ground springs

A ground spring may have spring characteristics in one or several degrees of freedom. The example below shows a ground spring with spring stiffness in x and y-directions. All other directions are fixed.



Support

Name : Sp10

Position : Point(18 m,5.499999999 m,4 m)

☒ Boundary Condition ☐ Boundary Stiffness Matrix

Boundary conditions:

☒ Let x change y and z

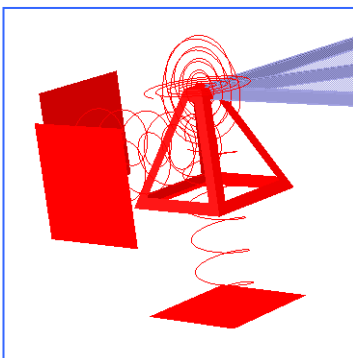
	Fixed	Free	Prescribed	Dependent	Super	Spring	Spring stiffness
x							100 [kN/m]
y							100 [kN/m]
z							0 kN/m [kN/m]

☒ Let rx change ry and rz

	Fixed	Free	Prescribed	Dependent	Super	Spring	Spring stiffness
rx							0 kN*m [kN*m]
ry							0 kN*m [kN*m]
rz							0 kN*m [kN*m]

5.3 Boundary stiffness matrix

It is also possible to define a ground spring with additional data. The previous option assumed a spring characteristic with no diagonal offset values. By using the boundary stiffness option, the offset values may be defined.



Support

Name : Sp6x6

Position : Point(27 m,0 m,4 m)

☐ Boundary Condition ☒ Boundary Stiffness Matrix

Stiffness Matrix:

	1	2	3	4	5	6	Boundary Type
1	10	0 kN/m	0 kN/m	0 KN	0 KN	0 KN	Spring
2	0 kN/m	100	0 kN/m	0 KN	0 KN	0 KN	Spring
3	0 kN/m	0 kN/m	100	0 KN	0 KN	0 KN	Spring
4	0 KN	0 KN	0 KN	1000	0 kN*m	0 kN*m	Spring
5	0 KN	0 KN	0 KN	0 kN*m	10000	0 kN*m	Spring
6	0 KN	0 KN	0 KN	0 kN*m	0 kN*m	10000	Spring

Fixed
Free
Prescribed
Dependent
Super
Spring

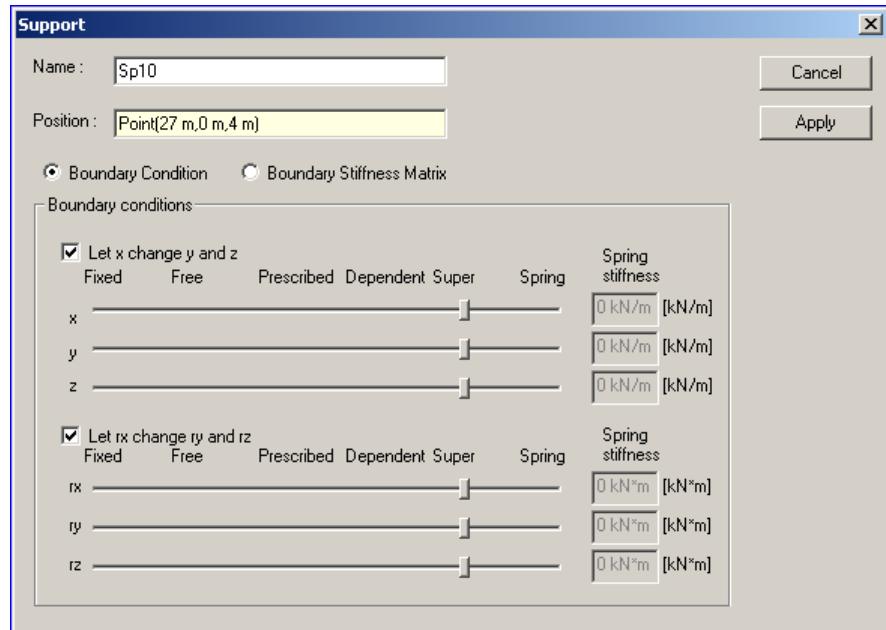
Symmetry is enforced, hence it is sufficient to specify the upper triangle of the matrix.

5.4 Create a super-element

When using boundary condition type “super”, a finite element model with connection points (or super-nodes) is created. The finite element model can be used in a super-element assembly generated by Presel.

Super-nodes are automatically created where there are boundary conditions of type “super” (at a point or along a curve).

Remember to specify superelement type (**Edit/Rules/Meshing**) prior to making the mesh. The finite element model can now be exported from the command **File/Export/FEM**.

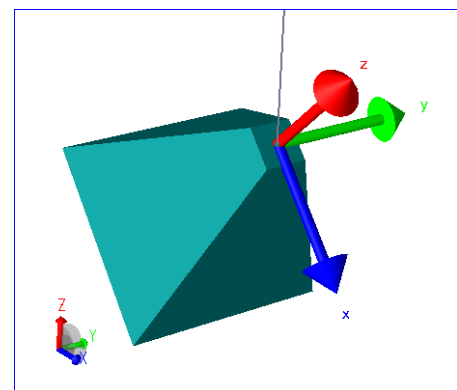
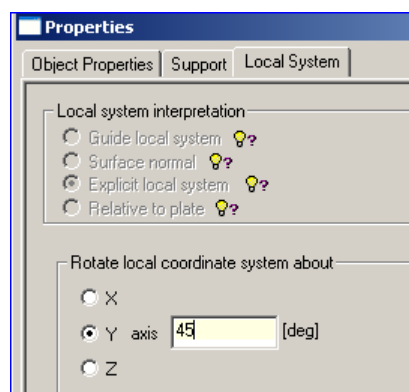
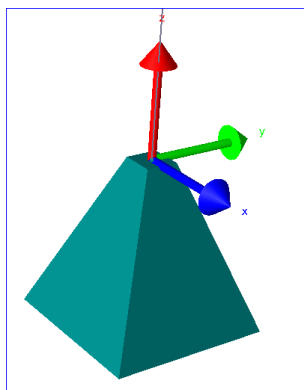


5.5 Local co-ordinate systems

Boundary conditions are specified according to a Cartesian co-ordinate system. The program default is the global co-ordinate system, it is however possible to apply a local co-ordinate system overriding the global system. The procedure is to define the values according to the local co-ordinate values and then rotate the boundary conditions so that the co-ordinate system is aligned with the desired orientation. There are two possible options

- Rotate around the local co-ordinate system and additionally relative to a plate/shell for curves
- Refer to a user defined co-ordinate system

The example below shows a boundary condition with its local coordinate system (select, **RMB, Labels, Local Coordinate System**), the rotation (select, **RMB, Properties, Local System**) and the new orientation as a result of a 45 degree rotation around local y co-ordinate.



The other option is to define a local co-ordinate system; select the boundary condition, **RMB**, *Properties, Local System*) as shown to the right.

In stead of the rotate option, select specify local coordinate system.

To define the local system as e.g. generated from a 45 degree rotation around the y-axis, the following data needs to be specified.

The input to the option “Local System” is `LocalSystem(Vector3d(1,0,-1), Vector3d(1,0,1))`. The data input in this case is for the local x and z axis (the y-axis is always known when x and z is given).

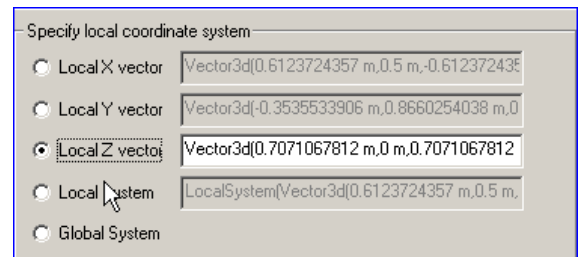
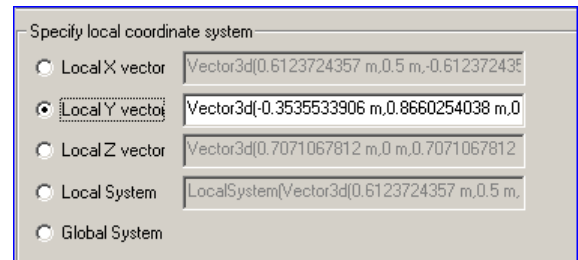
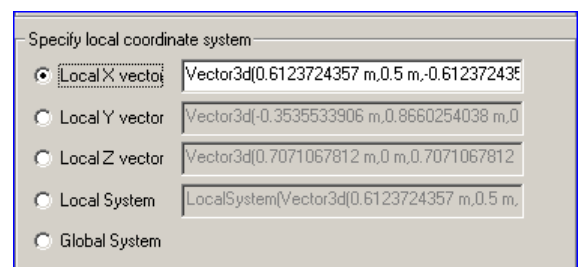
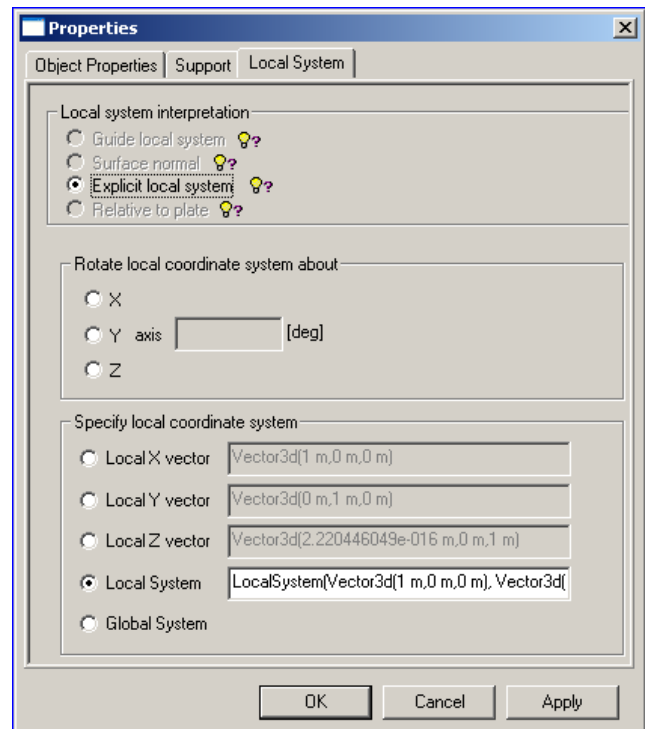
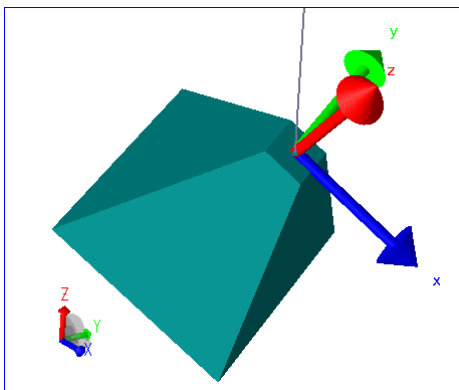
Alternatively, you can specify for individual x and z vectors:

Local x: `Vector3d(1,0,-1)`

Local y: `Vector3d(1,0,1)`

Remember to click apply for each update of the vectors.

The following example shows a local system including a 45 degree rotation around the y-axis followed by a 30 degree rotation around z-axis.



If you click “Global System” the local coordinate system will be re-aligned with the global system.

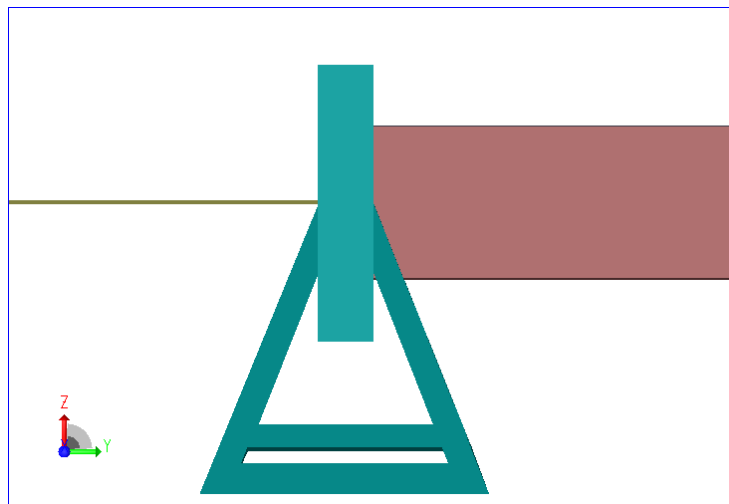
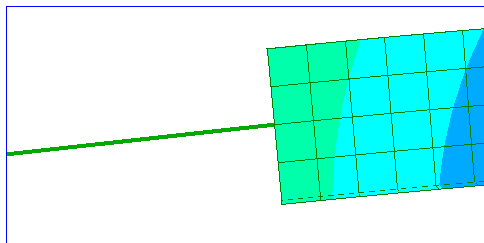
5.6 Rigid link support

A rigid link support is a connection between an independent point (also known as master node) and dependent points (also called slave nodes). The connection can contain various degrees of dependencies; typically all dependent nodes shall inherit both translation and rotation of the master node or only the translations.

Rigid link supports are often used to ensure a correct transition between a beam and shell model or at the ends of a cargo hold model.

The picture to the right shows a rigid link support between a beam end and the edges of the incoming plates.

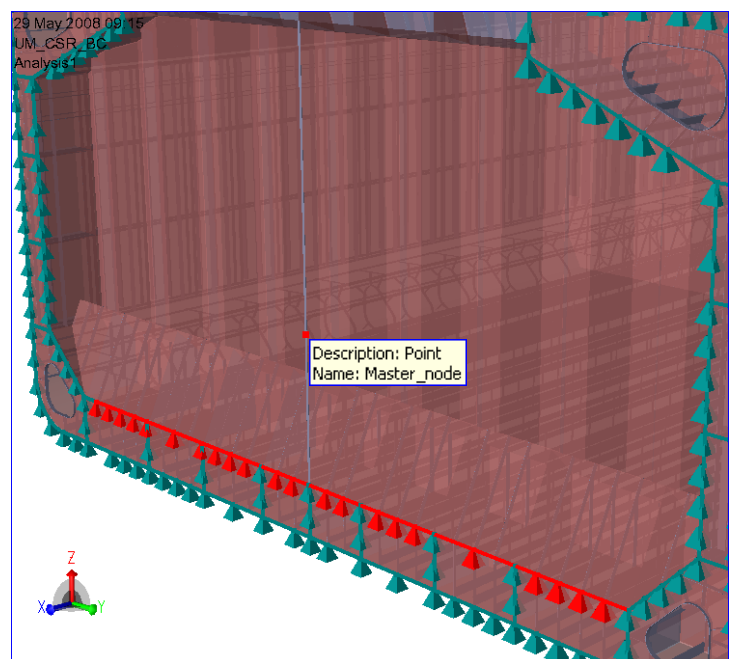
The rigid support link will ensure that the vertical plane (the plate edges) will remain flat when loads inducing rotations are applied to the model (rigid body motion).



In the example above loads are applied to the beam and deformations are applied to the plate edges as a result of the rigid link support. In this case the independent points (translation DOF) are fully coupled with the dependent point (the beam end). As can be seen, the plate edge remains flat where the finite element nodes along the plate edge are forced to move as a plane using the characteristics of the beam cross section.

For cargo hold analysis the dependent points may be partially coupled (some degrees of freedom) to the independent point to simulate the connection with the structure outside the cargo hold model. In this case the relevant cross section of the hull does not behave as a rigid body; the deflections and rotations depend on which degrees of freedom that are coupled between the master and slave nodes.

Typically, the slave nodes along the highlighted edge are connected for the translation degrees of freedom of the master node. When using GeniE together with the Nauticus Hull FEA template the boundary conditions as shown to the right are automatically applied.



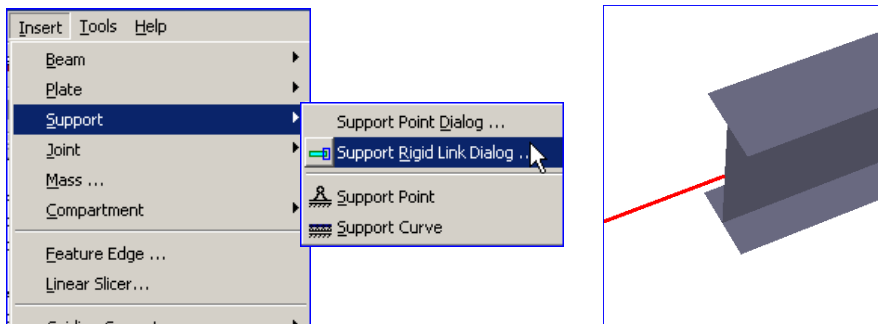
When all degrees of freedom are dependent between the master and the slave nodes (the first example above) all edges in a region is included. In the second example some of the degrees of freedom are dependent; in this case the relevant lines (by using support curves) are included. Both options are described in the following.

5.6.1 Rigid body behaviour

Rigid body behaviour (or “flat planes remain flat planes”) requires that all finite element nodes (slave nodes) in the plane are dependent on the translation degrees of freedom of a dependent point (master node). The procedure is to

- Decide the independent point (normally the beam end connected to shell structure)
- Decide the boundary conditions for the independent point (normally free in all degrees of freedom)
- Decide the dependent points (normally a volume including all free edges of the shells defining the flat plane)

A rigid support link is defined from **Insert/Support/Support Rigid Link Dialog**. A typical example will be used to explain how to define a rigid link support ensuring rigid body motion behaviour between shells and an incoming beam.



The dialogue includes important help (click the light bulbs to see them)

Insert Support Rigid Link

Name:

Independent point:

Boundary condition of independent point: _____

☒ Let x change y and z

	Fixed	Free	Prescribed	Dependent	Super	Spring	Spring stiffness
x	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0 kN/m [kN/m]
y	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0 kN/m [kN/m]
z	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0 kN/m [kN/m]

☒ Let rx change ry and rz

	Fixed	Free	Prescribed	Dependent	Super	Spring	Spring stiffness
rx	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0 kN*m [kN*m]
ry	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0 kN*m [kN*m]
rz	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	0 kN*m [kN*m]

Region of dependent points

☒ Include all edges in region

☐ Include only support points and curves in region

Box center:

Box extent X: [m]

Box extent Y: [m]

Box extent Z: [m]

☐ Local coordinate system orientation

OK Cancel Apply

Independent point

- Rigid link will define a region of the structure that will act as a rigid body. The feature can be used to force all nodes in a cross section to move as a plane giving it the characteristics of a beam cross section. The displacements of the nodes within the region is governed by a point defined as the independent point.

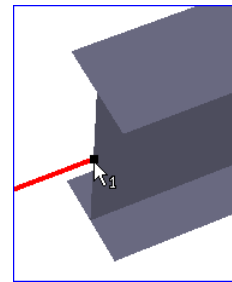
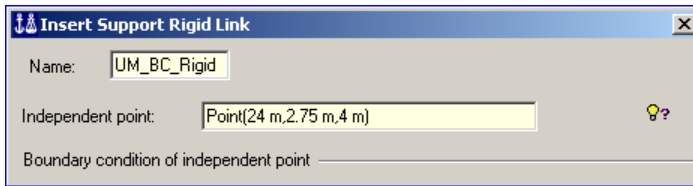
Dependent points

- If all edges in the region shall be included, the translational dofs are dependent and the rotational dofs are free for all edges.
- If only some degrees of freedom are to be dependent on the points in the region must be defined by support points or curves. Each degree of freedom of the supports must then be properly specified as dependent, free, fixed etc.

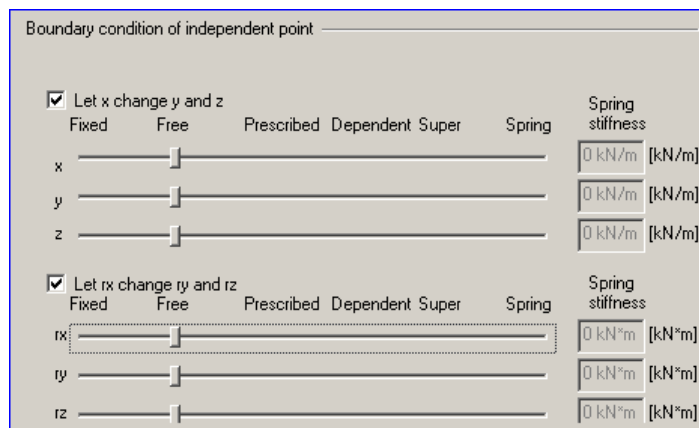
Orientation

- The local coordinate system orients the box region in space. The local system does not affect the orientation of the boundary conditions.

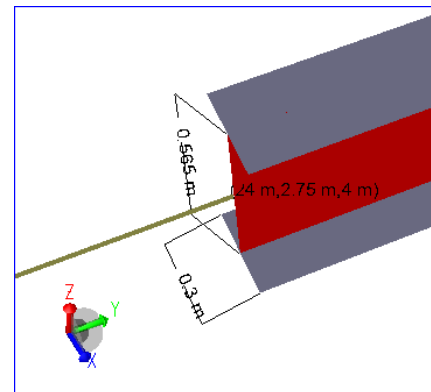
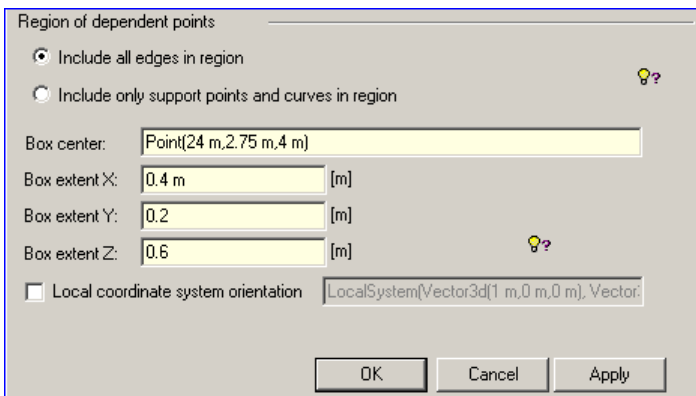
The first step is to define the independent point (the master node). Normally this is the end of the beam as shown to the right.



The second step is to define the boundary conditions for the master node. In this example these are set to free for all degrees of freedom, but there may be cases where some of the degrees of freedom have other constraints (e.g. fixed or spring).

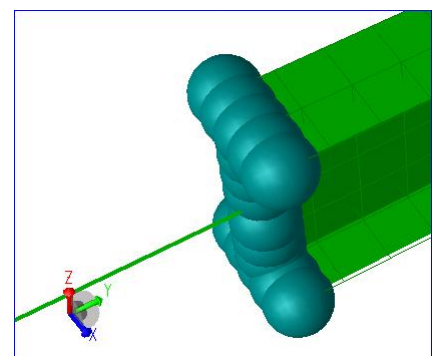


The third step is to specify the independent points (the slave nodes) by a volume definition. Define the centre of the volume and the extent in x, y and z direction. The default is global x, y and z, but it is possible to use a local co-ordinate system also. All finite element nodes inside the volume will become slaves of the master node.



The box centre in this case is the same as the master node (but it can be different) and the typical lengths in global x and z directions are 0.3 m and 0.565 m. When defining the box the extent values should be slightly larger than the typical lengths to include all edges. The box extent in y-direction is used to define the volume. If you only want the finite element nodes along the plate edges to be slave nodes you need to ensure that the box extent in y-direction does not include regions of the plates including additional finite element nodes.

When using the above option "All edges in region" all slave nodes will per default be dependent for the translation DOF and free for the rotational DOF.

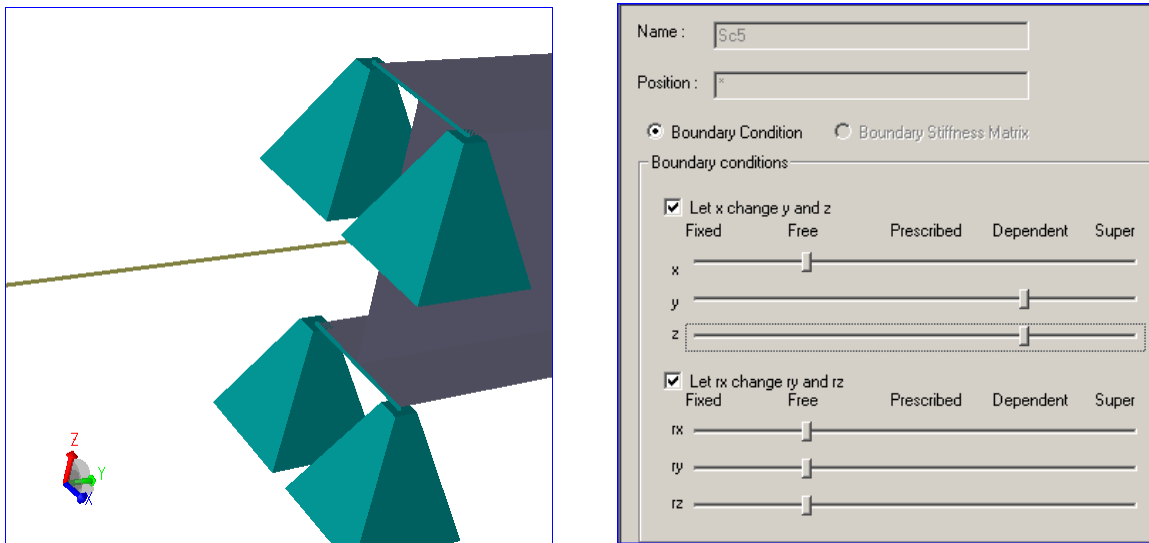


5.6.2 User defined behaviour

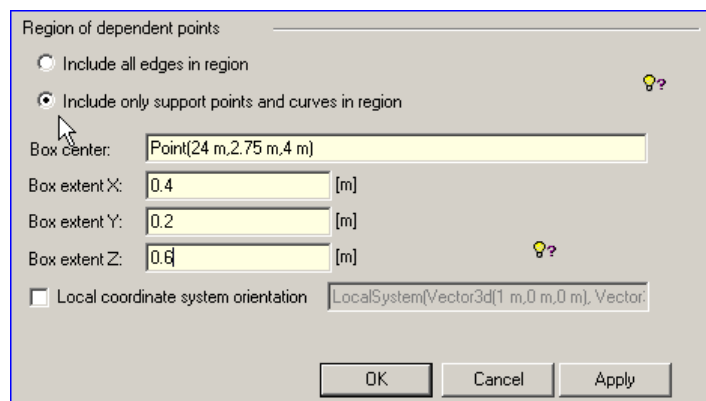
If you want to specify a user defined relation between the master and slave nodes the procedure is slightly different from the rigid body behaviour. In stead of selecting a volume leading to a system default definition of the dependencies (fixed in translations and free for rotations) individual support curves are used and dependencies may be applied to each of them. The slave nodes are defined by the extent of the support curves.

The same model is used to explain this alternative, please notice that support curves have been defined along the bottom and top flanges. In other words, there are no dependencies between the end of the beam and the edge of the web except for the single point of contact. This is not a realistic scenario, but it is used to explain the functionality.

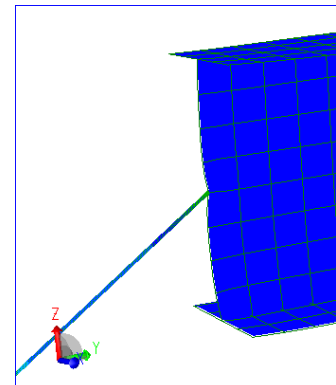
The support curves (defining the slave nodes) specify dependencies in y and z directions.



To ensure that the support curves (and support points) are selected you need to tick off the relevant option in the dialogue box.



As can be seen from the results to the right there is no dependencies between the incoming beam and the edge of the web except for the finite element connection itself. In this case the flat plain does not remain flat as for the rigid body motion behaviour.



6. MAKE AND CONTROL THE FINITE ELEMENT MESH

Prior to making a finite element model you should ensure that the model passes the geometry check from **Tools/Structure/Verify**. The finite element mesh depends on the settings you specify as well as the topology, the edges and the vertices of the model; see Sections 3.3.6 and 3.3.7 for more details.

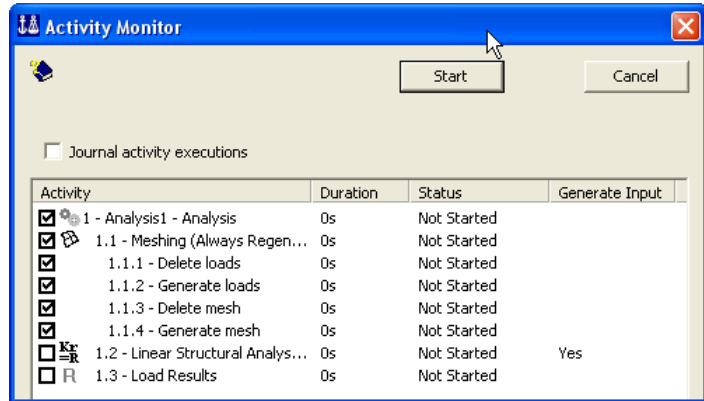
A finite element mesh is created from **ALT+M, Tools/Analysis/Create Mesh** or from **Tools/Analysis/Activity Monitor (ALT+D)** by deselecting the activities not needed.

For an explanation of the different meshing steps, Delete loads, Generate loads, Delete mesh, Generate mesh, see volume 1, chapter 3.13.2.

The following priorities are used when creating the finite element mesh.

1. Number of elements along a line (beam or feature edge)
2. Mesh density applied to a line (beam or feature edge) and a plate
3. Global settings specified in the mesh rules
4. If there are no settings, the settings specified along an edge are inherited

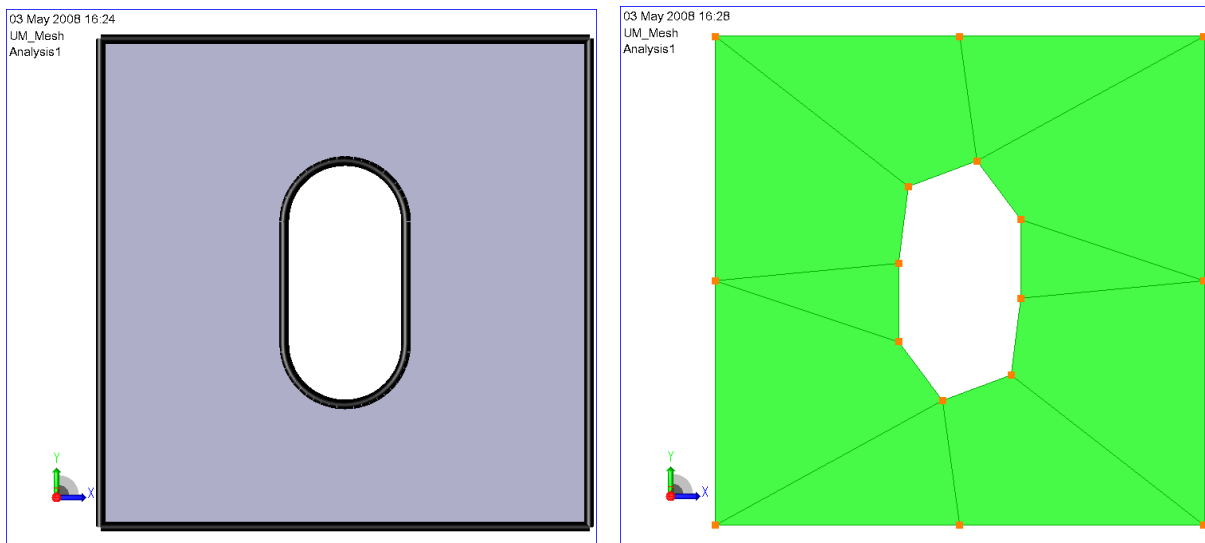
The various mesh settings are explained in the following with examples on the effect of the settings.



6.1 General

It is possible to make a finite element mesh without assigning any mesh properties to the model except for the program defaults (see next section). In this case the mesh will be generated based on the topology, the edges and the vertices of the model. Furthermore, the individual elements will be as large as possible.

The model below (a plate with a hole) shows the edges and the vertices. These are used to determine the mesh layout.



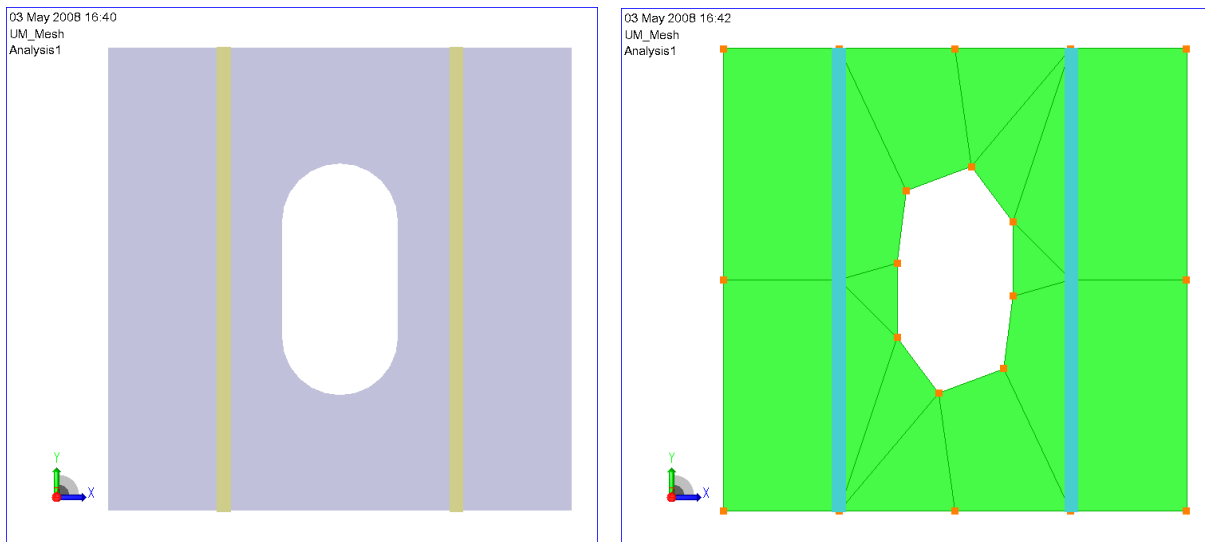
It is advised that you use a global mesh density (and set it to default) to give the program some guidance when making the mesh, see the next Section.

As can be seen the mesh is coarse and the results from a structural analysis will not be usable except for local details. It is therefore necessary to assign mesh control parameters globally to the entire model or locally to parts of the model.

The mesh layout is also controlled the more edges and vertices are built into the model. Such lines are defined when beams, plates and feature edges are added to the model.

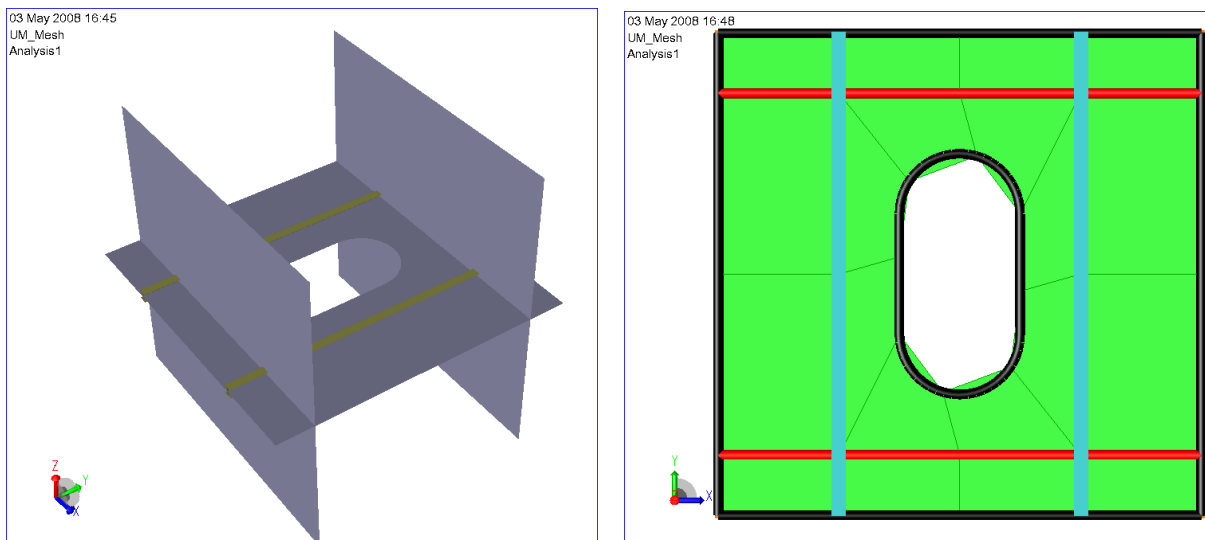
6.1.1 Refine mesh by inserting a beam

The example below shows the new mesh after inserting two beams.



6.1.2 Refine mesh by inserting a plate

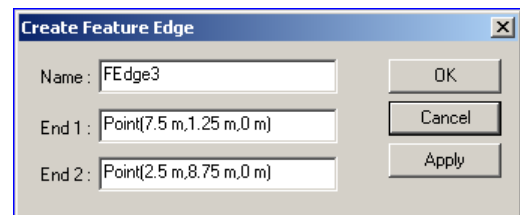
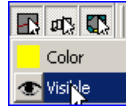
Two vertical plates are inserted below. This will add two new vertices in the model to control the mesh. These are shown in the mesh view (notice that this view shows the mesh of the horizontal plate only).



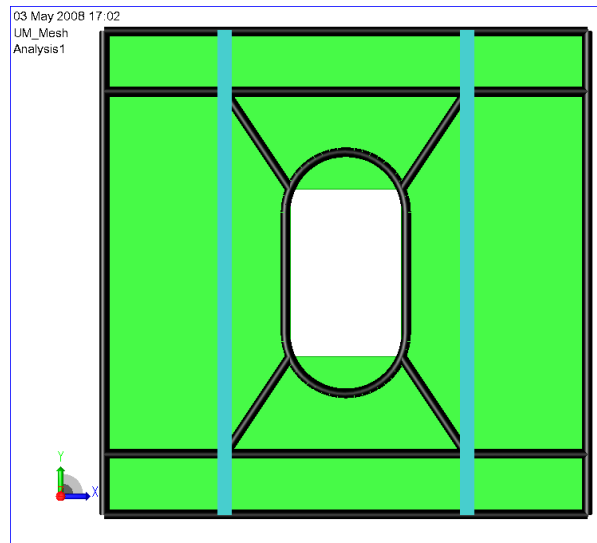
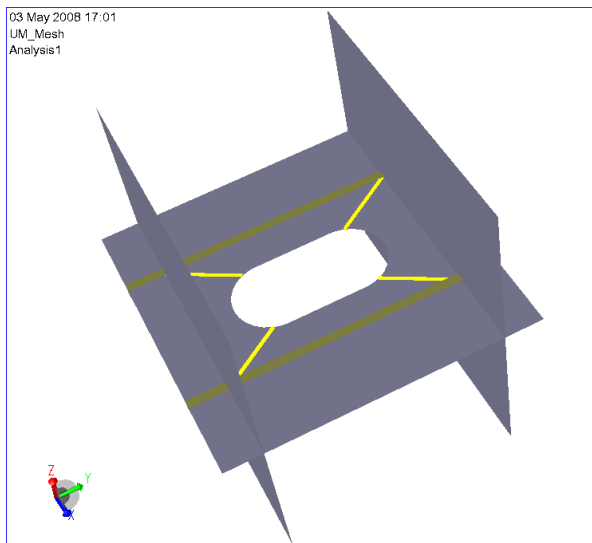
6.1.3 Refine mesh by inserting feature edges

It is also possible to add vertices to control the mesh; this is done by inserting feature edges (**Insert|Feature Edge**).

Remember to activate the tool button

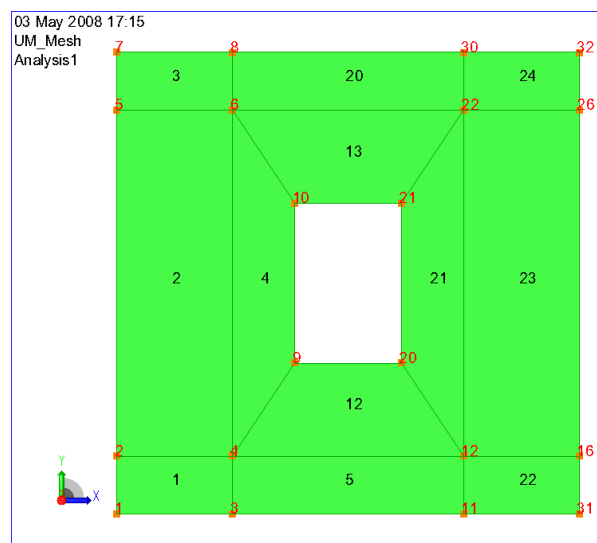
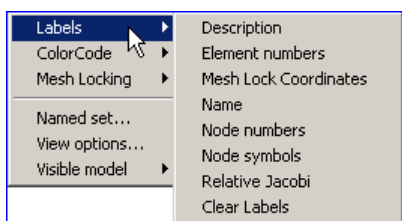


The model below contains two feature edges (in yellow colour). There are now enough vertices to make a regular mesh with high quality, but not for detailed stress analysis since the mesh density is too coarse.



6.1.4 Labelling

To label a mesh entity, select the finite element mesh (entire mesh or parts of it), click **RMB** and select *Labels*. The picture below shows the mesh including element numbers (black colour), node numbers (red colour) and node symbols (orange colour).



6.1.5 Documenting the finite element mesh

The quality of the analysis results depends on the quality of the finite element mesh. In addition to the users experience there are two ways to assess the quality of the finite element mesh:

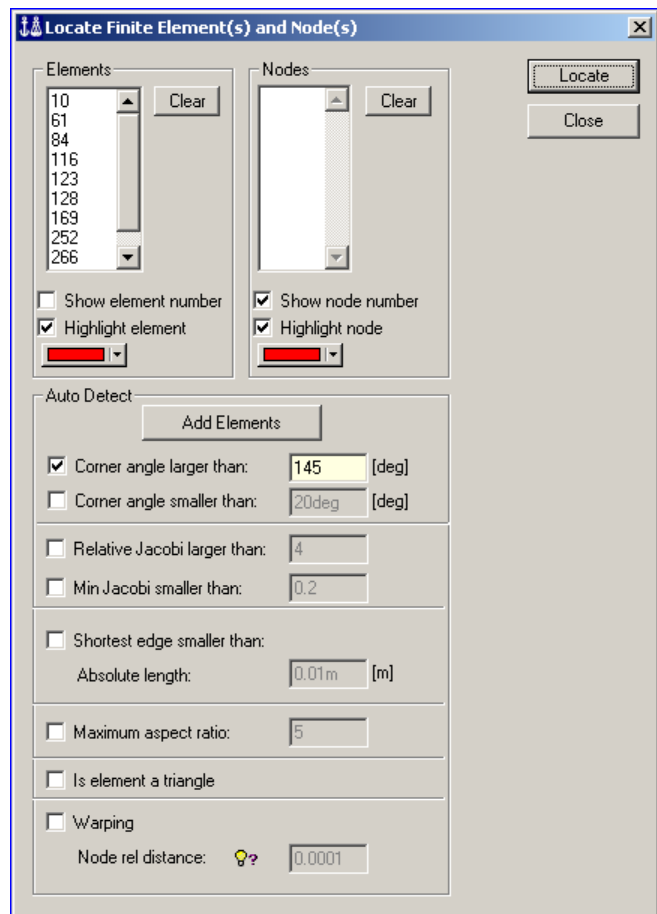
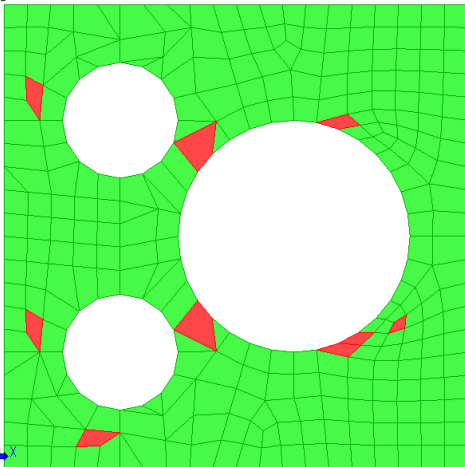
- Check certain parameters like mesh angles, the Jacobi determinant, edges, aspect ratios and warping – how to document is shown below.
- Look at stress resultants to see that there are no abrupt changes in the contour plot – this is an indication that the finite element mesh is too coarse.

The **Tools/Analysis/Locate FE** will help you to document the mesh. In the example below all finite elements with a mesh angle higher than 145 degrees are highlighted.

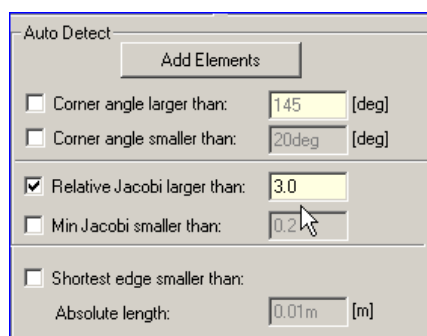
The procedure to highlight these elements is to:

- 1) Specify maximum angle (in this case 145 deg)
- 2) Click on *Add Elements*
- 3) Click on *Locate*
- 4) The elements should now be highlighted in red colour – if you don't see any colour coding click anywhere in the graphic window

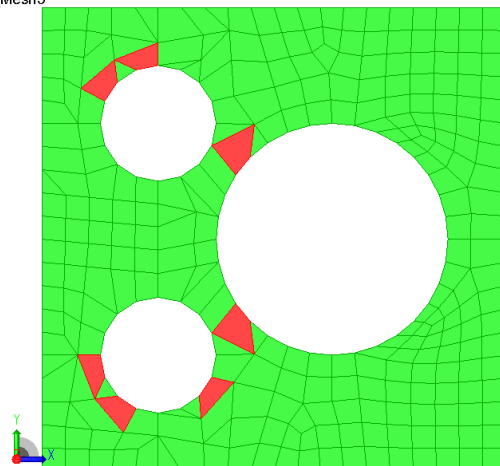
05 May 2008 03:04
UM_Mesh5



Similarly, the picture to the right shows those finite elements with a relative Jacobi larger than 3.0.

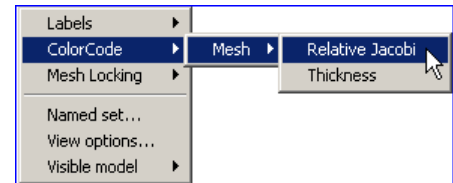


05 May 2008 03:06
UM_Mesh5

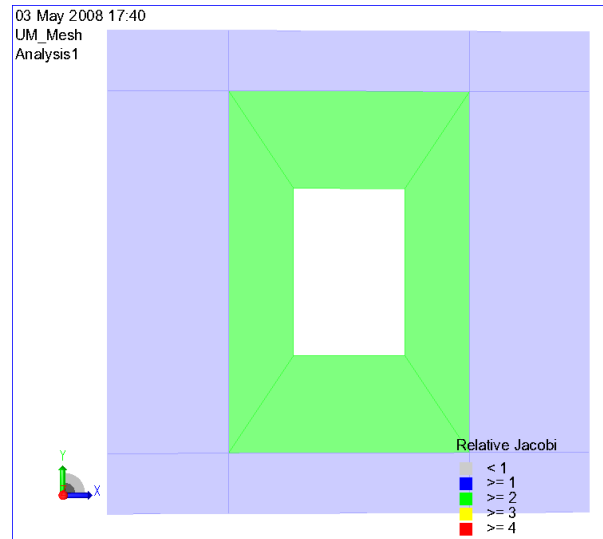
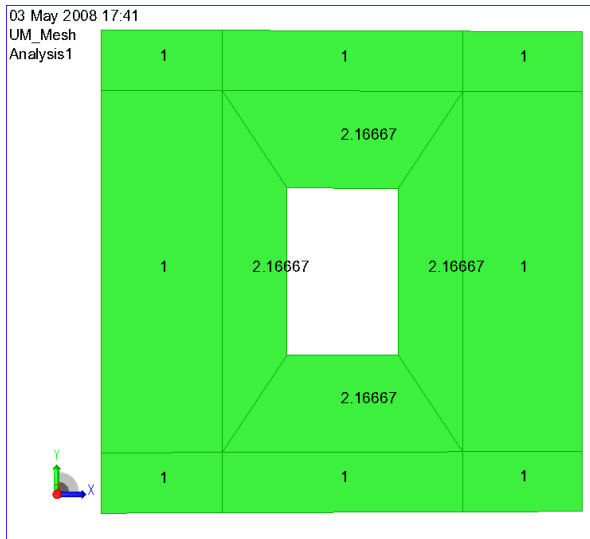


6.1.6 Colour coding

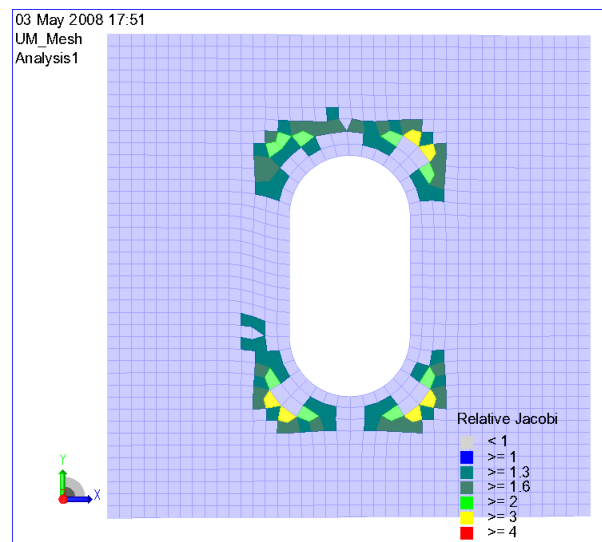
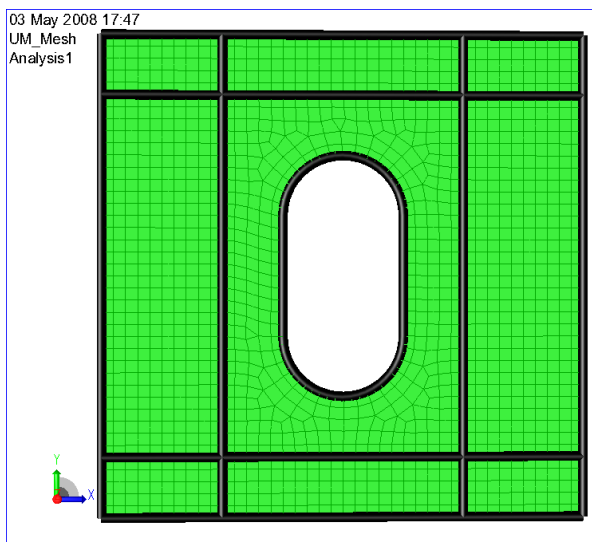
Colour coding of the mesh is used to verify the relative Jacobi determinant or the actual plate thickness used in the finite element analysis. The plate mesh thickness is the same as the plate thickness except when corrosion addition is used (see next Section). To do colour coding, select the mesh, click **RMB** and select *ColorCode*.



The example below shows the relative Jacobi for the horizontal plate (colour coding as well as labels shown).



The rest of this Chapter explains how to control and improve the mesh so that it can be used to derive reliable stresses and displacements depending on the type of analysis to be performed, typically global analysis, local stress analysis or fatigue analysis.



Chapter 7 *Make analysis models and run analysis* explains how to make different analysis models (or finite element models) from the same concept model.

6.1.7 Element types

GeniE can create both 1st and 2nd order elements when creating a finite element model. You choose which element formulation to use either from the rules setting for mesh generation *Edit/Rules/Meshing*.

The following finite element mesh types are generated and exported to the FEM file. More details about each element type may be found in Sestra User Manual.

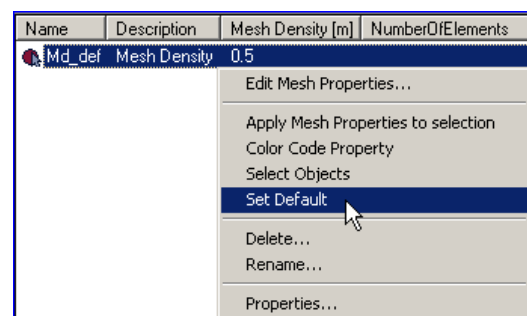
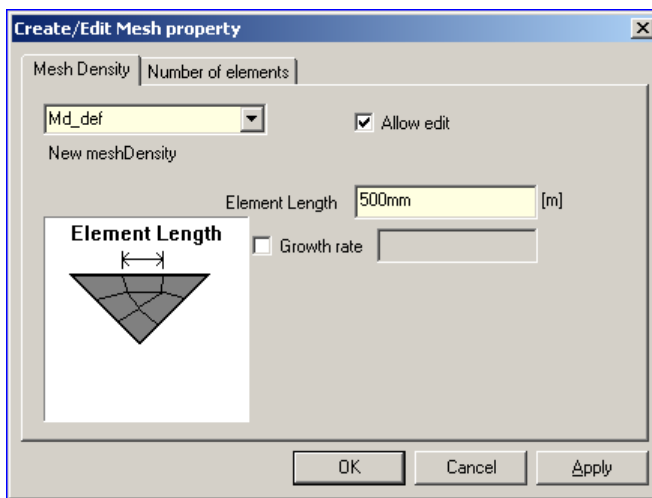
Name	Type	Order	Comments
2-node beam element	BEAS	1 st	
3-node beam element	BTSS	2 nd	Straight beams can be used in code checking
Quadrilateral flat thin shell element	FTRS	1 st	
Triangular flat thin shell element	FQUS	1 st	Inserted when adjusting mesh rules to split elements
Quadrilateral sub parametric curved thick shell element	SCQS	2 nd	
Triangular sub parametric curved thick shell element	SCTS	2 nd	Inserted when adjusting mesh rules to split elements
Quadrilateral flat thin shell with drilling dof	FQAS	1 st	Includes the rotational dof around the axis perpendicular to the membrane in the membrane formulation
Triangular flat thin shell with drilling dof	FTAS	1 st	- o -
Non-structural 2 node beam element	BEAS	1 st	Special variant of BEAS with no contribution of the structural stiffness
Truss element	TESS	1 st	Element type with no bending stiffness
Spring to ground	GSPR	1 st & 2 nd	Includes the 6x6 matrix
Shim element	GLSH	1 st & 2 nd	Special variant of the 2 node spring element with equal stiffness in two translation directions. No stiffness in other directions.
One node mass element	GMAS	1 st & 2 nd	May be eccentric if connected to a finite element node with 6 dof.

Within the same model it is not possible to have both 1st and 2nd order element types.

6.2 Global mesh settings

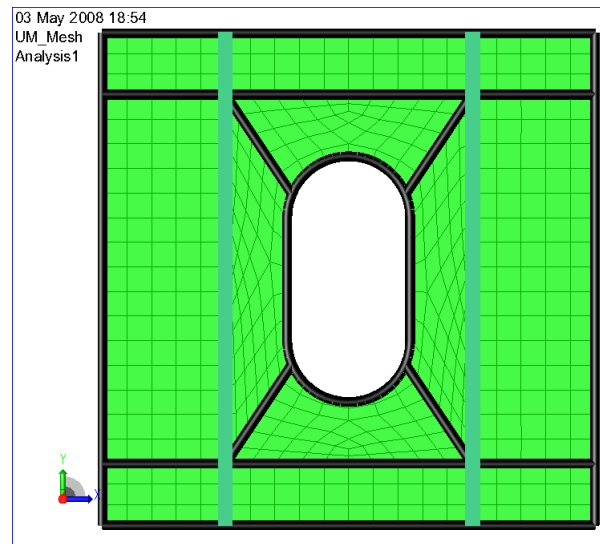
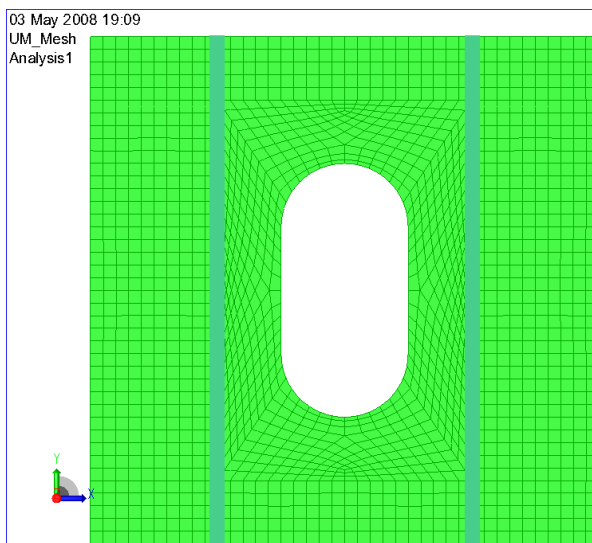
Global mesh settings are applied to the complete structure while local mesh settings are applied to parts of the model (see next Section). The various mesh settings should be used depending on what type of analysis to be performed as well as the degree of quality that is needed. There are two types of global mesh settings; mesh density and mesh parameters.

The mesh shown in the previous Section was made without any mesh density settings. To make the mesh less coarse (decrease the mesh size) a global mesh density can be applied. A mesh density is defined from **Edit/Properties/Mesh Property** or from the browser directly. In this case a mesh density of 500mm is specified. The mesh density can be assigned to individual objects or set global. The global mesh setting is over-ridden by local settings.



The new mesh with global mesh settings applied is shown to the right (horizontal plate shown). All other mesh settings are in accordance with program defaults.

Modifying the global mesh density to 250mm gives the mesh depicted below.



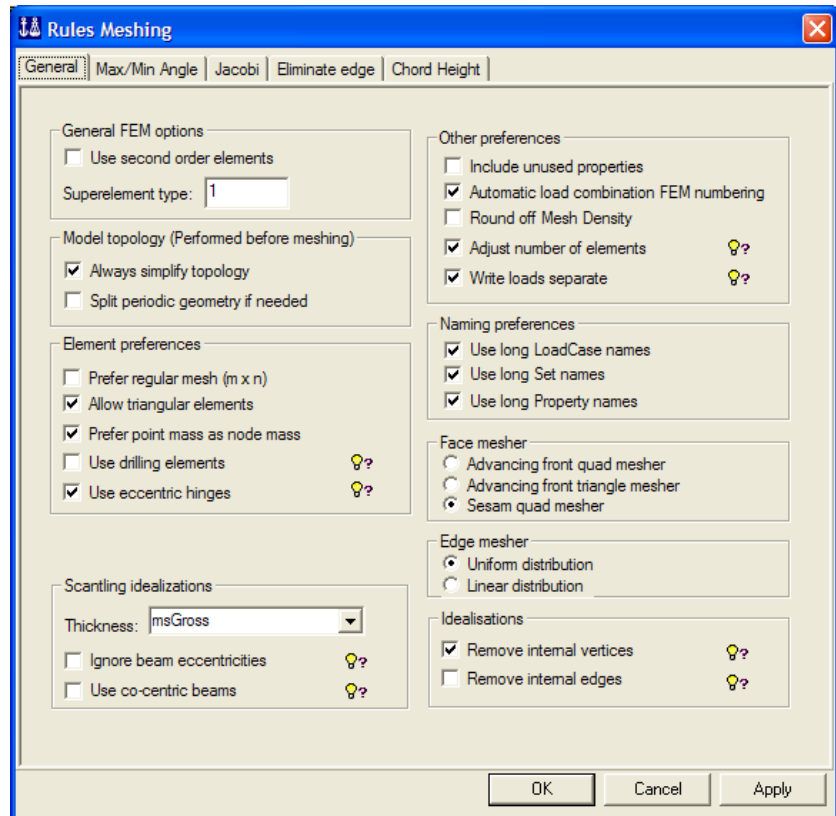
It is advised that you use a global mesh density (and set it to default) to give the program some guidance when making the mesh.

The global mesh parameters can be edited from **Edit/Rules/Meshing**. The rules specified herein are global (they apply to the entire model); you may override these for local part of the model using the features for local mesh settings (see following Section).

The mesh rules consist of five different parts (General, Max/Min, Jacobi, Eliminate edge and Chord Height).

Each of these options is described in the following.

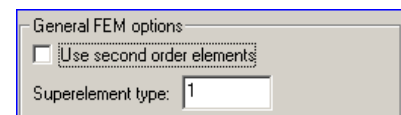
The program default settings are shown to the right.



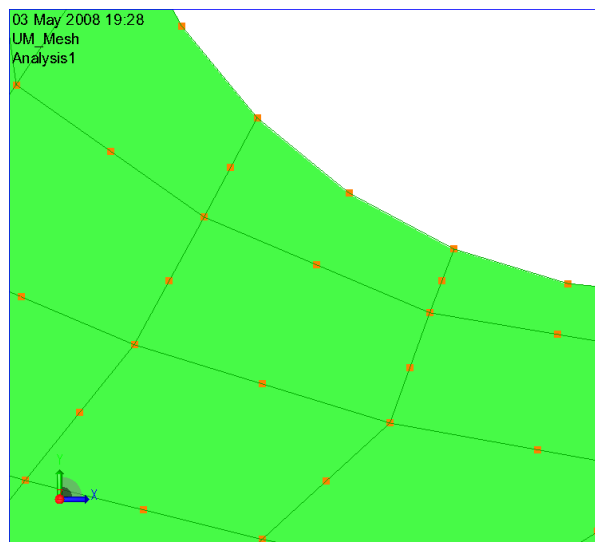
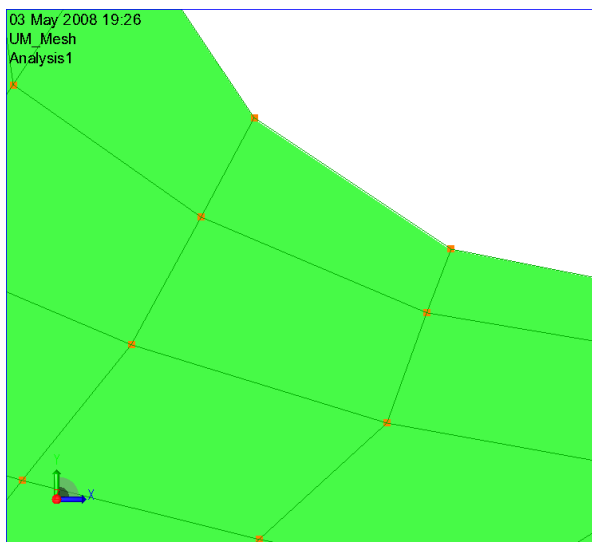
6.2.1 Mesh settings - general

6.2.1.1 General FEM options

This option is used to specify which superelement number that shall be used when creating the finite element model. This may be relevant if you want to make several analysis models from the same concept model or make superelements for use in an assembly.

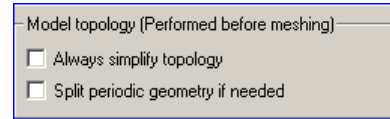


By default first order finite elements are used. When you tick off for second order elements the finite elements will change to typically 8 nodes per quadrilateral plate and 3 nodes for beams. The types of elements supported are shown on the next page.

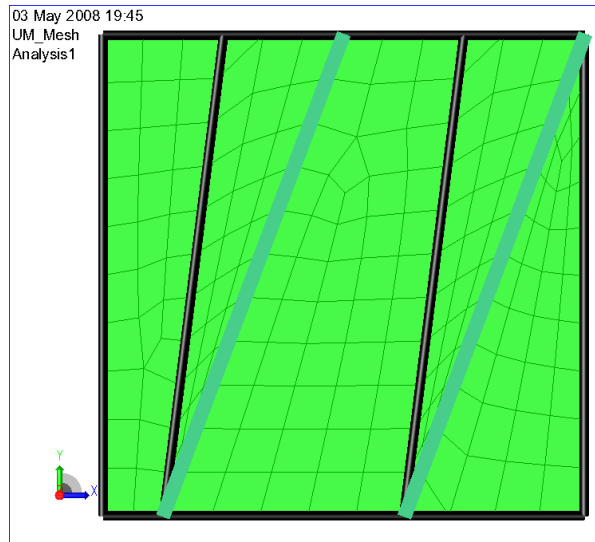


6.2.1.2 Model topology

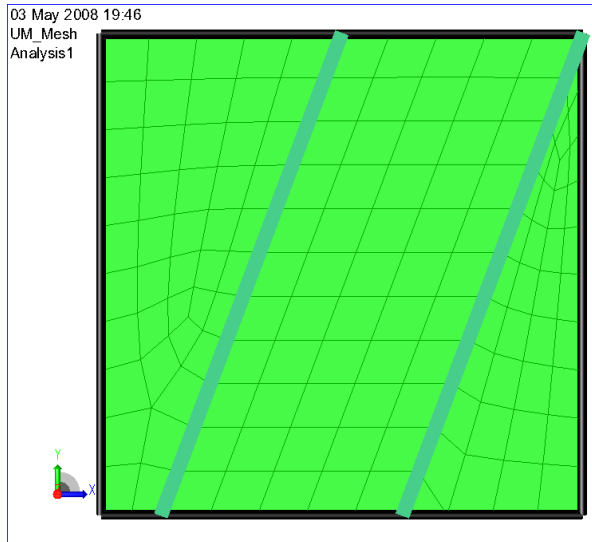
The option for always simplify topology before meshing cleans up the model prior so that no un-necessary vertices or edges are part of the model when meshing takes place. This can have tremendous effect on the mesh that is created and **you are advised to always use this option.**



The example below shows meshes that is created on the same model with and without simplify topology. When making the model two beams are moved without cleaning up the model.

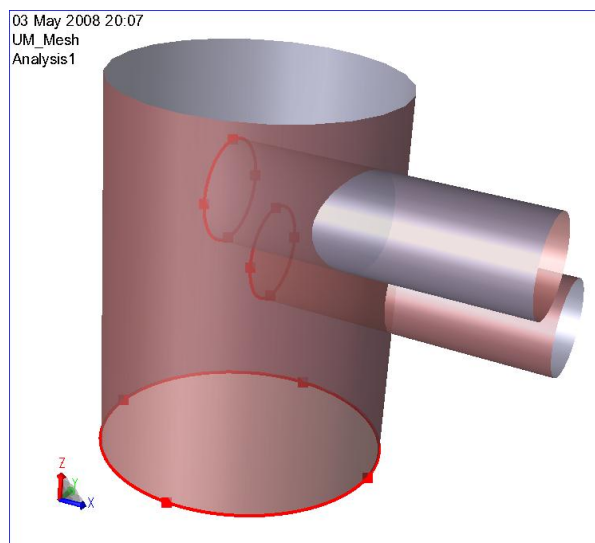


Mesh without simplify topology
No internal edges removed

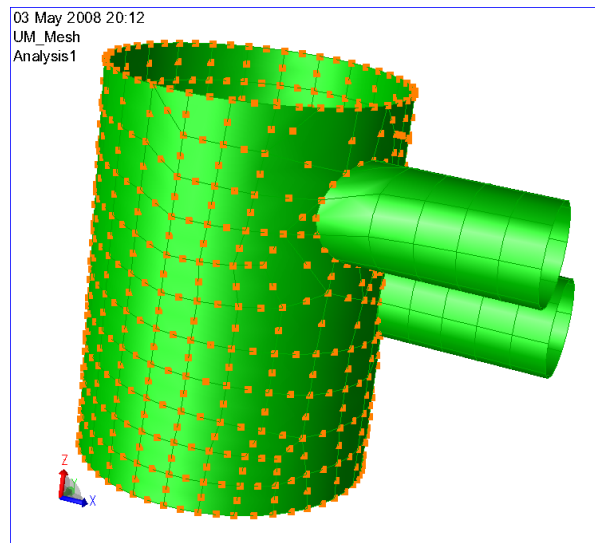


Mesh with automatic simplify topology
Internal edges not needed are removed

Tubular surfaces may be made from a sweeping operation where a 360 degree guideline is used as reference (highlighted below). In some cases GeniE is not able to make a finite element mesh on such surfaces. In these rare events, you should use the option “Split periodic geometry if needed” to insert an edge along the surface in the longitudinal direction.



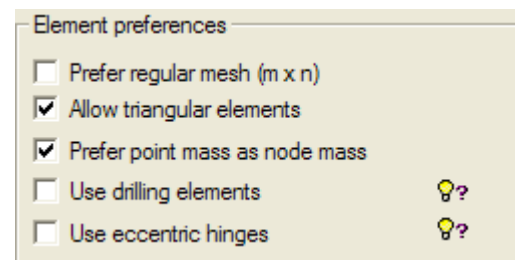
360° guidelines used in sweep operations



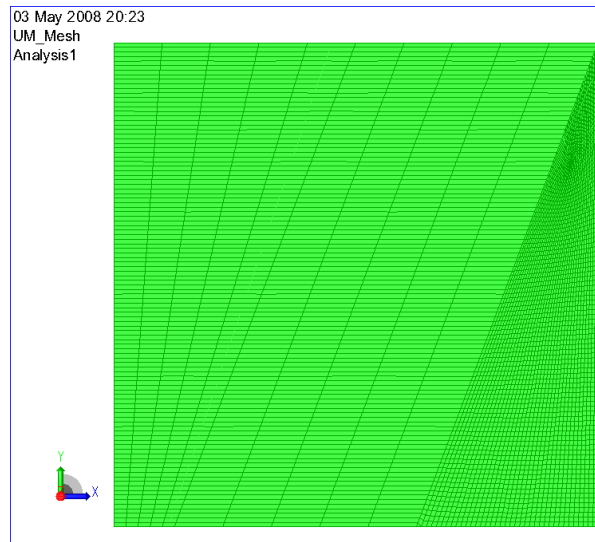
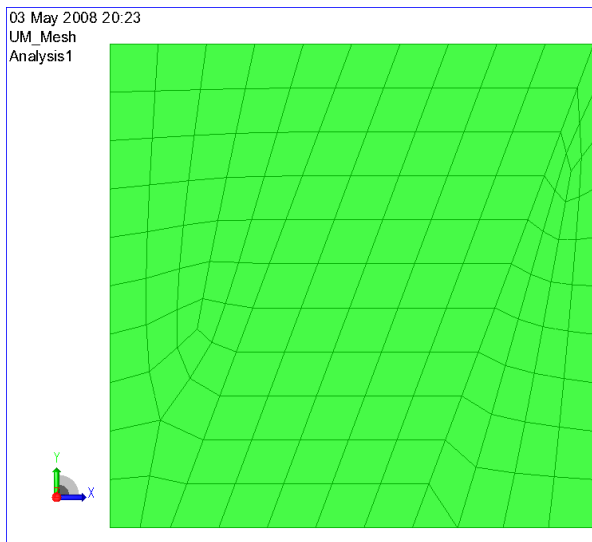
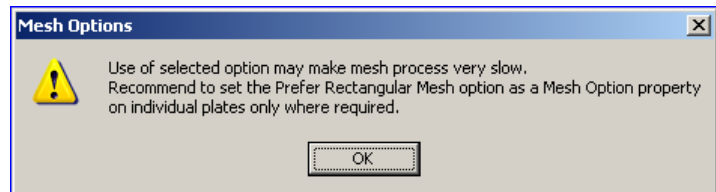
Mesh automatically created

6.2.1.3 Element preferences

The option for “Prefer rectangular mesh” should be used with care as it will enforce rectangular mesh. In order to do so it may be necessary to use small element sizes to achieve such. This option should be used on regular and rectangular structures only.



When selecting this option you will see a warning as follows:



The meshing philosophy in GeniE is to always make a mesh. In order to do so it may be necessary to use triangular elements. Disabling the “Allow triangular mesh” will ensure that as few as possible triangular elements are used to create the mesh. As such it is not an option for allowing triangular elements or not.

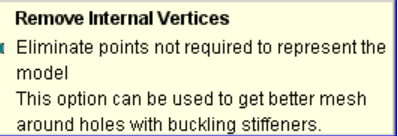
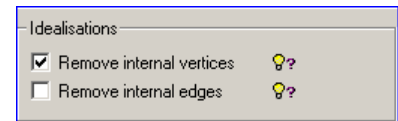
The parameter “Prefer point mass as node mass” will ensure that a mass element with zero length will be written to the FEM file. If deselected an eccentric mass element will be used.

If you tick off the “Drilling element” a special variant of the thin shell element is used. It has independent membrane and plate bending parts and utilizes the rotational dof's around the axis perpendicular to the membrane plane in the membrane formulation. The membrane part of the drilling element (FQAS) is composed of four triangular elements. The bending part is the same as for a thin shell element (FQUS). It is thus more stable for deformed shapes than the thin shell element for flat surfaces where membrane behaviour dominates.

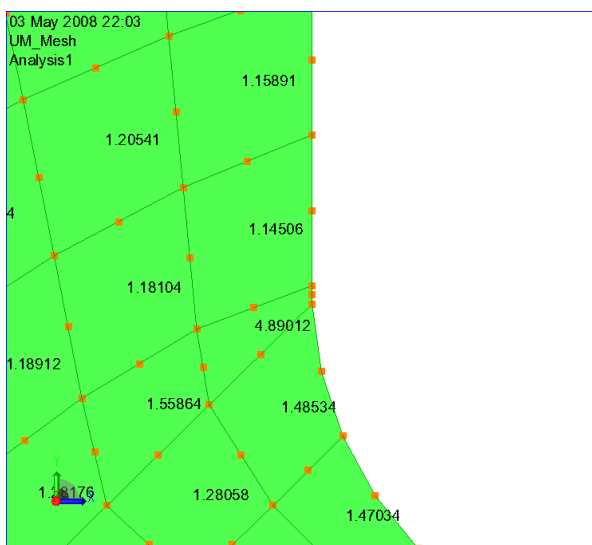
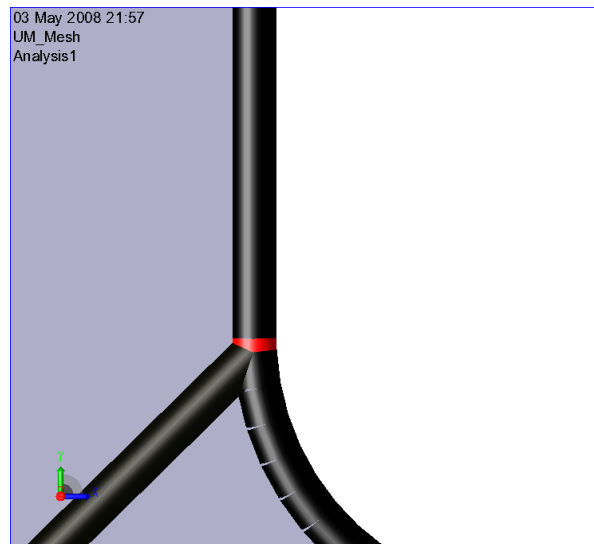
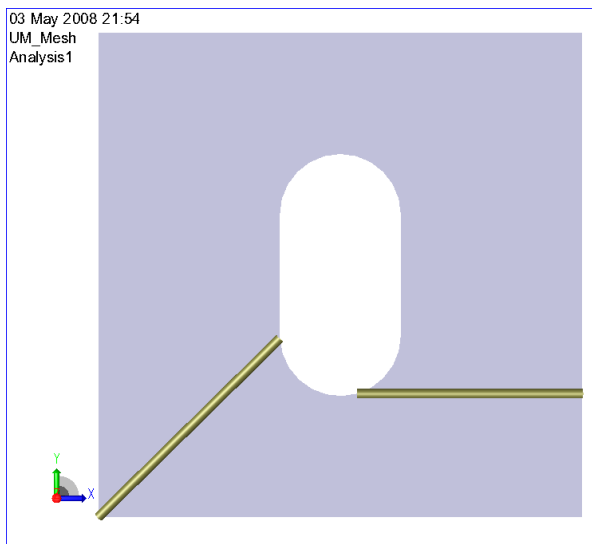
In order to get a more correct moment diagram for the beams the “Use eccentric hinges” may be checked. With the option checked hinges are moved from the node to the eccentric beam ends. This feature requires Sestra 8.4-04 or later.

6.2.1.4 Idealisations

The mesh is among others determined of the vertices and their points and it may be necessary to idealize the model prior to meshing to achieve a decent mesh. A typical problem may be where stiffeners do not intersect with the ends of vertices used to describe the shape of a hole or a structural component. In such cases the “Remove internal vertices” may be used to improve the mesh layout. This option is system default as it prevents the creation of distorted quadrilaterals into elements with large angles (often close to 180 degrees).

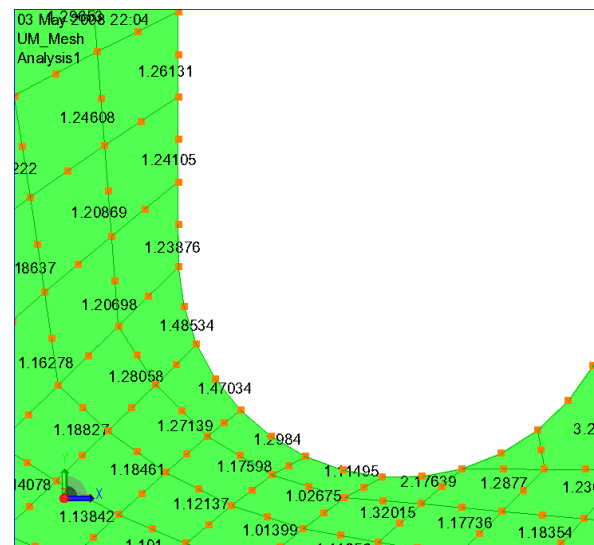


The effect of such idealisation is shown using a plate with a hole and two stiffeners. The stiffeners do not intersect with the ends of the vertices describing the curved part of the hole. There is a short distance from the intersection to the vertex end; this is highlighted below. As a consequence a very short side of an element is generated. The shorter the length of this line is, the worse the finite element becomes. The idealisation “Remove internal vertices” moves the incoming beam to the nearest end of a vertex used to describe the hole.



No idealisation

Jacobi determinant 4.89



Idealisation performed

Significantly improved mesh

Comparable Jacobi determinant 1.24

The “Remove Internal Edges” is used when you want to eliminate edges not required to make the mesh.

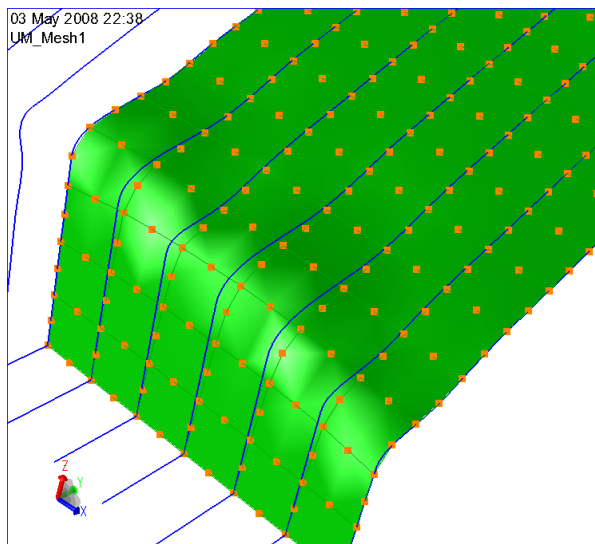
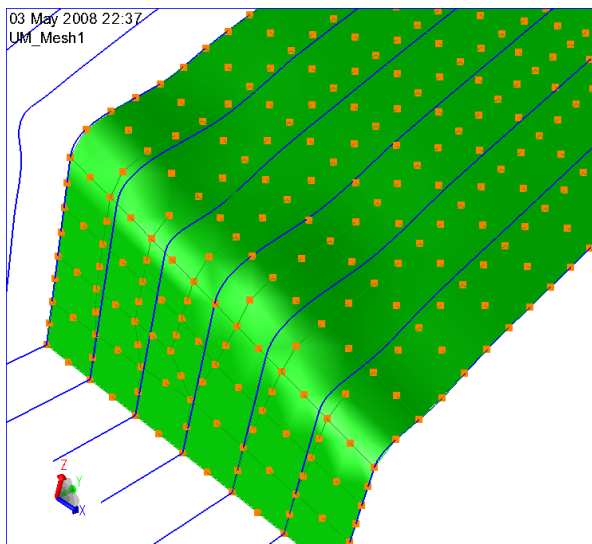
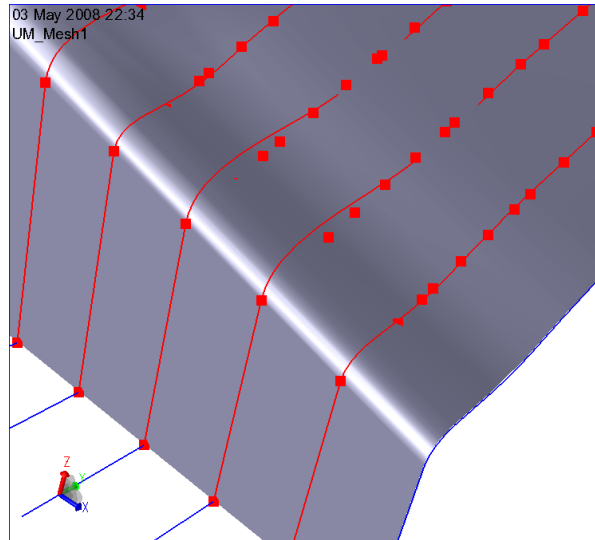
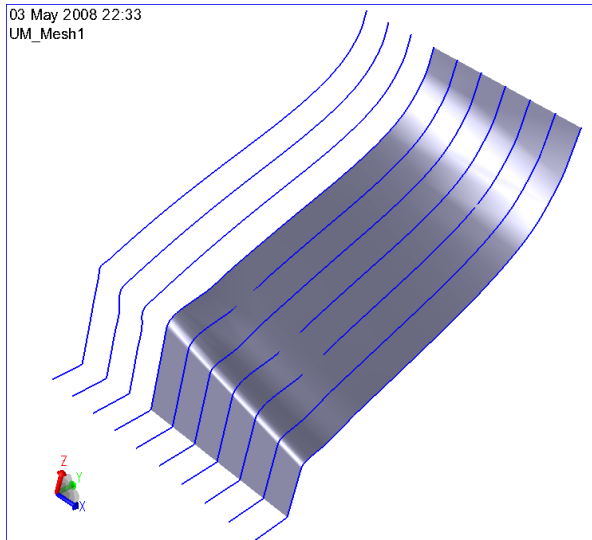
This option can be used when a model contains many internal edges as a result of importing data from external sources or when a complex model has many edges that are not needed in a finite element model (i.e. the finite element model does not have the same of accuracy as the model itself – like for example in a panel model).

Remove Internal Edges

- Eliminate edges not required to represent the model.

This option can be used to reduce the number of patches on a plate that needs to be meshed separately.

In the example below data from an offset table has been imported. The number of points for each imported vertice is high which may lead to edges between the vertices.



No idealisation
Mesh follows all edges part of the model
Mesh “identical” to structure
May lead to complex mesh details

Idealisation of internal edges
Mesh independent of internal edges
Mesh somewhat different than structure
Regular mesh

6.2.1.5 Corrosion addition

The Genie model is normally created using gross thicknesses, but thicknesses may be automatically adjusted to take corrosion into account according to the given context.

The transition from a gross thickness into a net thickness is not simple, it will depend on:

- the set of rules used
- the structure part in question
- the context in which net thickness will be used

In general, we assume

$$t_{\text{net}} = t_{\text{gross}} - t_{\text{corr}}$$

where

$$t_{\text{corr}} = t_{\text{cd}} \text{ where } t_{\text{cd}} \text{ is the sum of corrosion additions applied to each side of the plate/stiffener.}$$

For plates, we assume that the corrosion addition is specified for each side of the plate:

$$t_{\text{cd}} = t_{\text{c1}} + t_{\text{c2}}$$

where t_{c1} and t_{c2} are the corrosion additions applied to each side of the plate

For stiffeners, also one-sided corrosion additions are applied.

$$t_{\text{cd}} = 2 * t_{\text{c1}}$$

where t_{c1} is the corrosion addition applied to the stiffener

For pipes the outer diameter is retained. For all profiles the main dimension (i.e. height, width and diameter) are not modified, only the wall thickness for pipes and web/flange thicknesses for other sections is modified. For hollow profiles, it is assumed that corrosion addition is applied to both inside and outside.

The rules have different ways of rounding off corrosion additions, and Genie supports the needs of CSR Bulk, CSR Tank and DNV 1A1 rules.

Generic expression

$$t_{\text{corr}} = \text{fac} * \max (t_{\text{minimum}}, \text{Round}(t_{\text{cd}}, t_{\text{increment}}) + t_{\text{reserve}})$$

where

fac is a context dependent reduction factor, may be different when creating FEM mesh and performing capacity checks.

t_{minimum} is a specified "minimum" corrosion additions.

$t_{\text{increment}}$ is a step in thickness used for rounding values off

t_{reserve} is an specified "reserve" thickness which may be added

The table below gives the constants in the generic expression for the different set of rules

Rule	CSR Bulk, Plate	CSR Bulk, Stiffener	CSR Tank	DNV 1A1	General
fac (FEM)	0.5	0.5	0.5 (<i>1</i>)	1.0	1.0
fac (Capacity)	1.0	1.0	1.0	1.0	1.0
t _{minimum}	2.0 mm	0.0 (<i>2</i>)	0.0 mm	0.0 mm	0.0 mm
t _{increment}	0.5 mm	0.5 mm	0.5 mm	0.0 mm	0.0 mm
t _{reserve}	0.5 mm	0.5 mm	0.0 mm	0.0 mm	0.0 mm
rounding	Up	Up	Nearest	None	None

(*1*) For CSR tank: the f (FEM) as default be 0.5, but It should be remembered that fac (FEM) should be set to 1.0 for the fine mesh area of local models and fac (FEM) should be set to 0.25 for fatigue models.

For CSR tank: Corrosion addition from tables are given as sum of additions from each side. The user may split the rule value this so that 50 % is applied to each side of the plate.

(*2*) For CSR bulk : No minimum for internal stiffeners.

For CSR bulk : Corrosion addition from tables are given for each side of the plate. The user assigned these values to the relevant side(s) of the plates in Genie.

6.2.1.6 Corrosion addition – CSR Bulk example

The thickness correction t_c according to the CSR bulk ship rule is:

$$t_{cd} = \max[2\text{mm}, \text{Roundup05}(t_{c1} + t_{c2}) + 0.5\text{mm}]$$

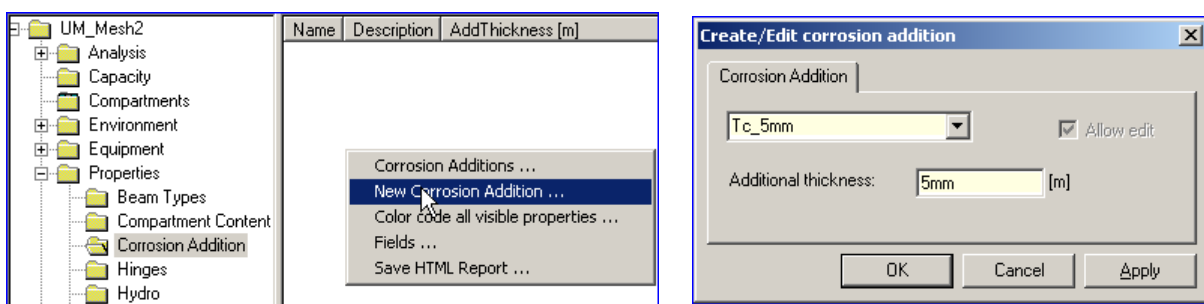
where

t_{c1} is correction addition on the front plate side (positive surface normal side)

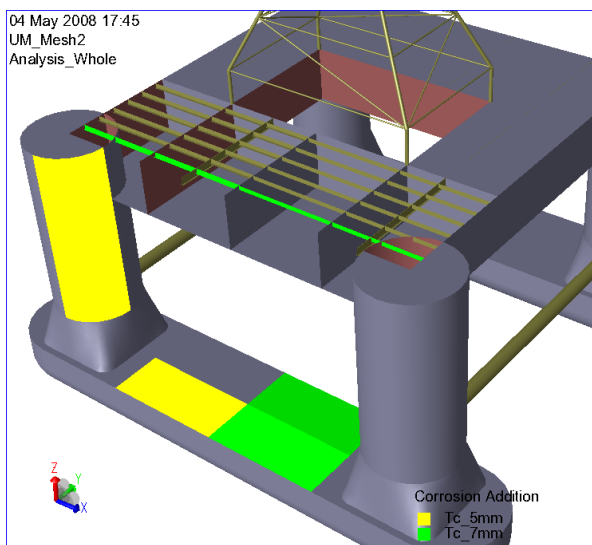
t_{c2} is correction addition on the back plate side (negative surface normal side)

Roundup05 means round up to nearest 0.5mm (e.g $t_{c1} + t_{c2} = 4.2\text{mm}$ then use 4.5mm)

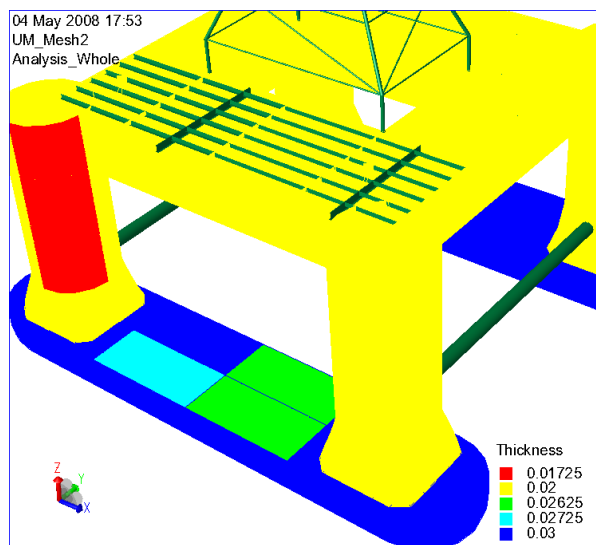
Corrosion additions are defined from **Edit/Properties/Corrosion Addition**, the browser or from compartment properties (see next Sections).



In the example below the corrosion additions $Tc_5\text{mm}$ and $Tc_7\text{mm}$ have been applied to some selected plates and stiffeners (outside side only).



Corrosion additions applied



Net scantlings as used in analysis
(select mesh, **RMB** and click colour code thickness)

In the example above corrosion addition of 7mm is applied to a plate with thickness 30mm. The net thickness is

$$t_{\text{net}} = t_{\text{gross}} - 0.5 * t_c = 30\text{mm} - 0.5 * t_c = 30\text{mm} - 0.5 * 7.5\text{mm} = 30\text{mm} - 0.3725\text{mm} = 26.25\text{mm}.$$

where

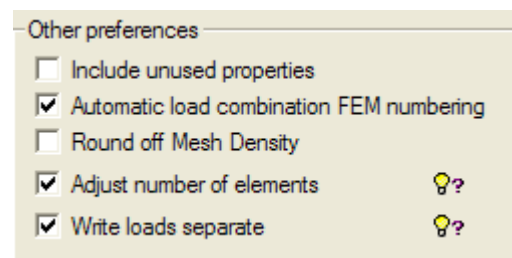
$$t_c = \max[2\text{mm}, \text{Roundup05}(t_{c1} + t_{c2}) + 0.5\text{mm}] = \max[2\text{mm}, \text{Roundup05}(7\text{mm} + 0\text{mm}) + 0.5\text{mm}]$$

$$t_c = \max[2\text{mm}, 7\text{mm} + 0.5\text{mm}] = 7.5\text{mm}$$

The web and flange thickness of the selected beam is also reduced with the thickness correction $t_c = 7.5\text{mm}$. To verify this you need to edit the FEM file. A new section property will automatically be created for the beams that have corrosion additions.

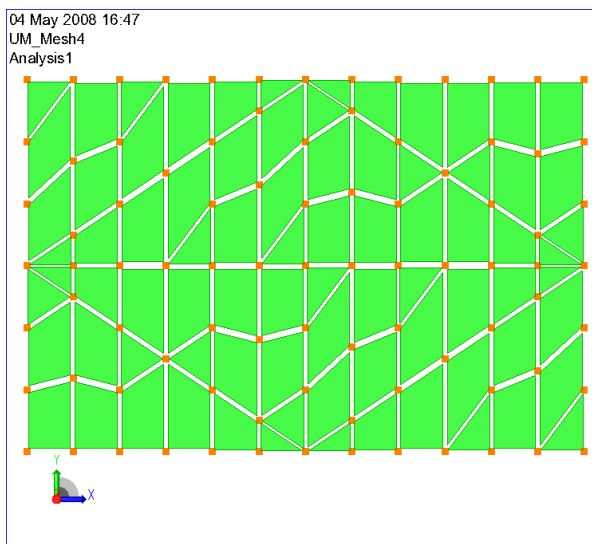
6.2.1.7 Other preferences

When you make a finite element model only the applied (those used in analysis) properties are written to the FEM file. If you want to add the unused properties, then you should activate the “Include unused properties”.



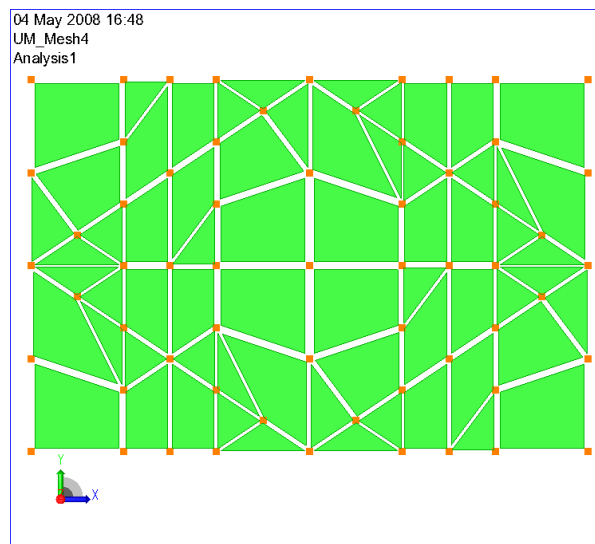
When you make a load combination, the finite element numbers will be in accordance with the default settings, i.e. the finite element loadcase numbers for load combinations will always start after the highest number for a basic load case or a wave load case (from Wajac). In case you want to control the finite element loadcase numbering manually you should de-select the option “Automatic load combination FEM numbering”.

GeniE will ensure that the maximum mesh densities are according to or smaller than the mesh settings applied to the model. In some cases this may lead to a more dense mesh than you desire. In such cases you can instruct GeniE to work with approximate mesh density settings, i.e. open up for finite elements having a slightly larger size than the maximum size specified by you. To do this you need to tick off the “Round off Mesh Density”.



Mesh density specified 2.0m.

No finite elements are larger than this size since the *Round off Mesh Density* is not activated.



Mesh density specified 2.0m.

There are elements larger than size 2.0 (but close to) since the *Round off Mesh Density* is activated.

The automatic “Adjust number of elements” along edges should normally be enabled (default), as the face meshes mostly will look better (with fewer triangles). Disable the option during problem solving if meshing hangs or if a poor mesh is observed, and check if this has any positive effect.

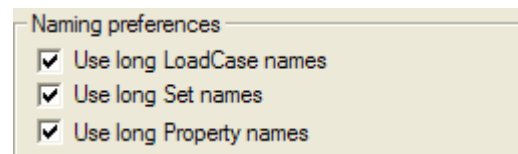
“Write loads separate” should normally be enabled (default), since memory usage is reduced. Disable the option during problem solving if some loads appear to be missing on the FEM model, and check if this has any positive effect.

6.2.1.8 Naming preferences

The default option is to make long names in GeniE and use these during post-processing inside GeniE (look at stresses, displacements, forces and code checking) or Xtract.

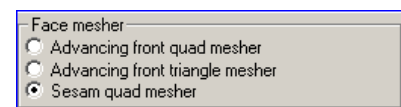
The result file (the SIN file) that is created during an analysis can also be used by other Sesam programs like Framework, Platemwork, Stofat and Cutres. These programs can read load case and set names up to eight characters long. To ensure compatibility with these programs you should use the default option of “Short names”. A long name will now be truncated to eight characters in upper case.

In case you specify ”Use long names” the above mentioned programs will refer to these using a number same as the number on the FEM file (loadcase finite element number and set finite element number).



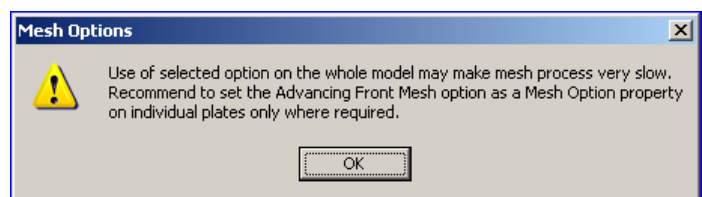
6.2.1.9 Face mesher

There are three options for selecting the face mesher. “The Sesam quad mesher” is targeting slender and regular structures while the “Advancing front mesher” focuses more complex shapes. It is possible to use both options for the same model; to do this you should use local mesh settings as explained in the next Section.

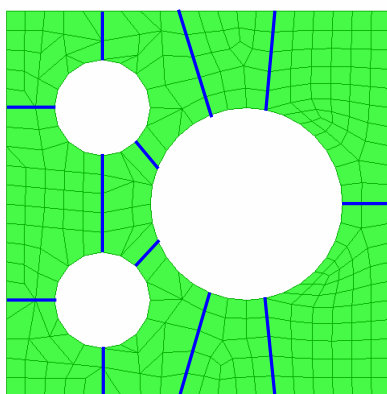


The “Sesam quad mesher” divides surfaces into patches and creates a mesh based on these. The “Advancing front mesher” generates the mesh along the edges first before filling the rest of the surface. This means that the “Sesam quad mesher” normally gives the best mesh in the middle of a surface (or patch) while the “Advancing front mesher” generates best quality mesh along the outer boundaries (or edges) of a surface.

The “Sesam quad mesher” should be used for structures without complex details like holes and complex intersections between surfaces. For detailed analysis of complex structures the “Advancing front mesher” should be used. Observe that the latter option can result in longer time to make the mesh and hence a warning is given when selecting this option.

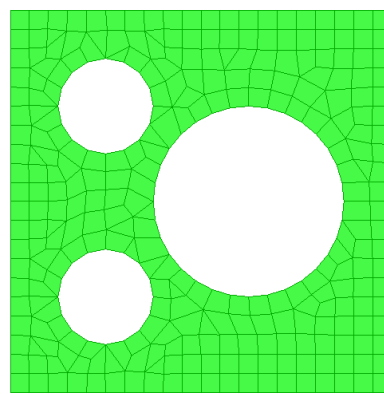


04 May 2008 19:19
UM_Mesh5

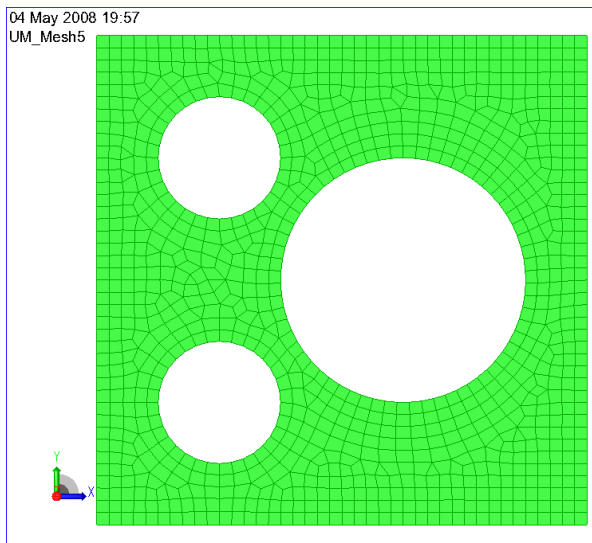


Sesam quad mesher

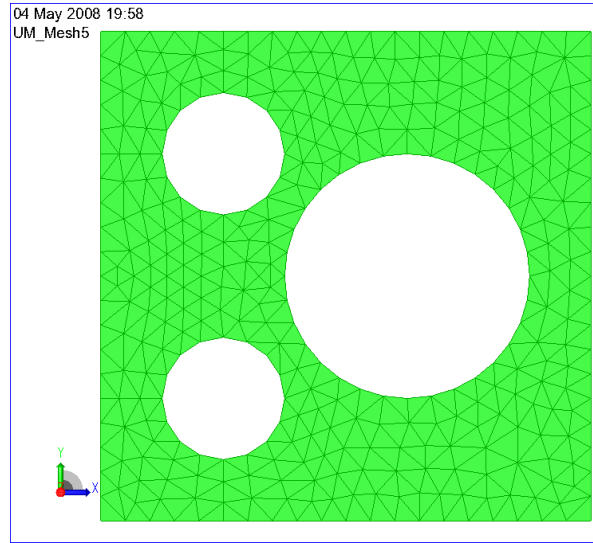
04 May 2008 19:28
UM_Mesh5



Advancing front quad mesher

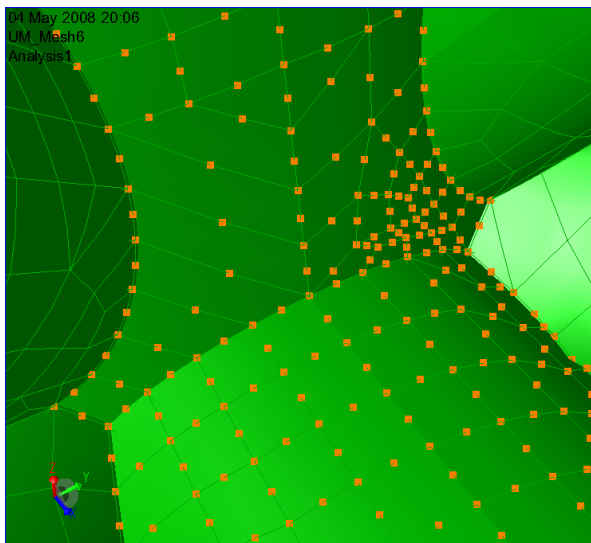


Increased mesh density improves mesh quality significantly when using *Advancing front mesher*

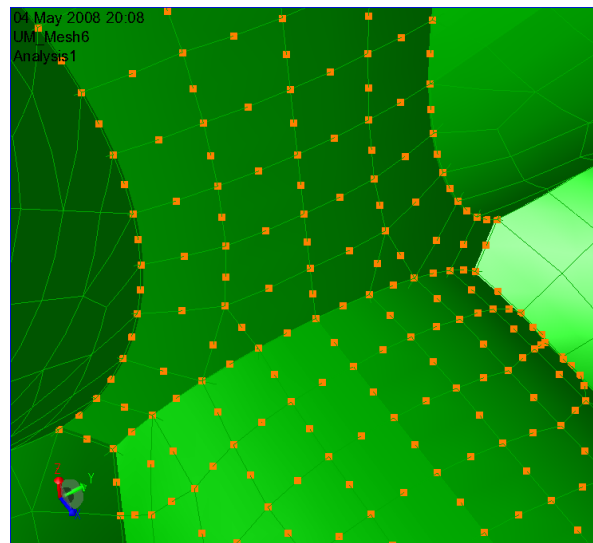


Example on *Advancing front triangle mesher*

The effect of improved mesh quality can also be shown using an overlapping tubular joint:

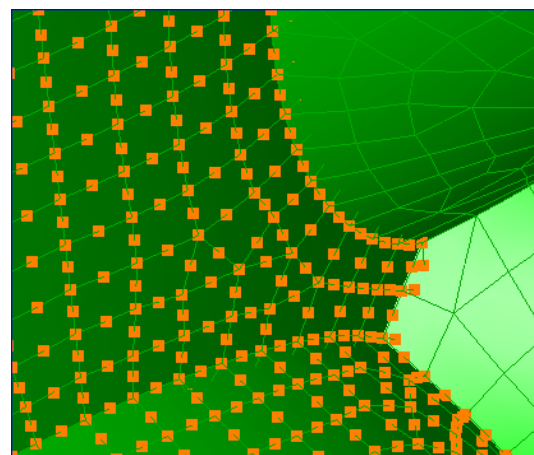


Sesam quad mesher
uses patches to create mesh



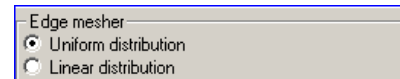
Advancing front quad mesher
makes mesh along edges first

Reducing the mesh size (in this case 1/2 size of above) leads to a highly regular mesh yielding quality results using the *Advancing front quad mesher*.

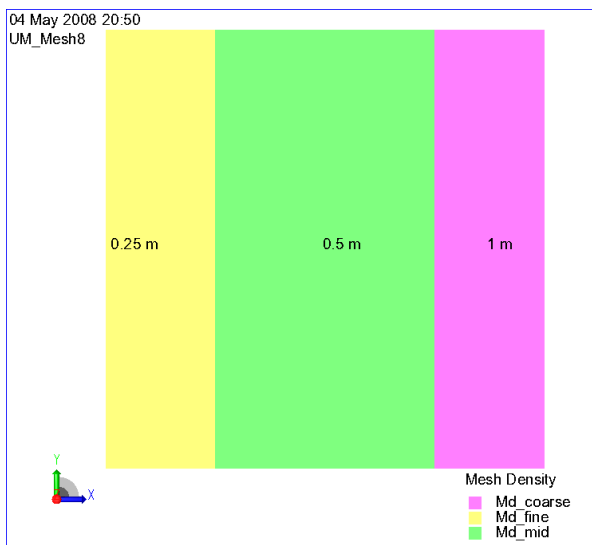
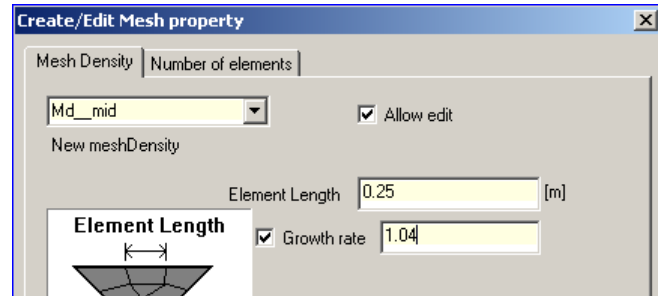


6.2.1.10 Edge mesher

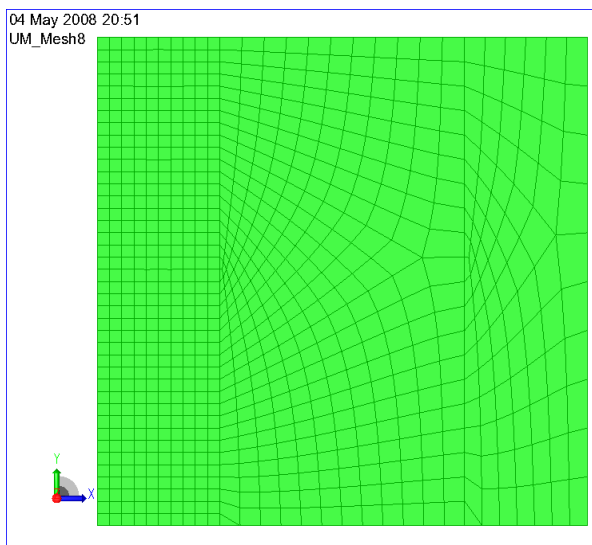
It is possible to specify the mesh growth rate along edges when the mesh densities vary (typically from coarse mesh to fine mesh).



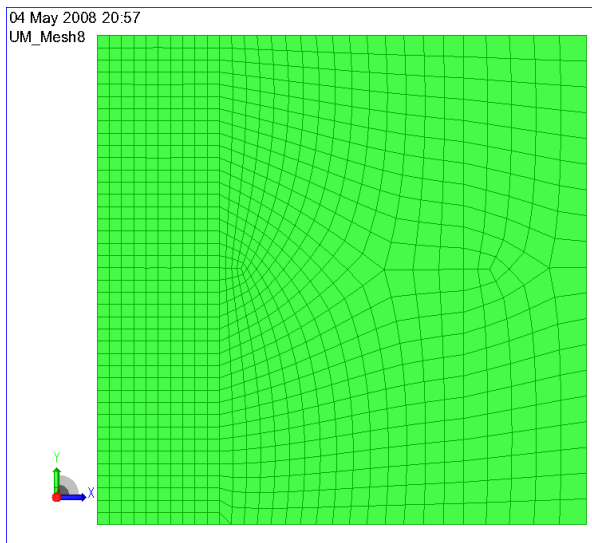
When the default option “Uniform distribution” is used, GeniE will seek to create the mesh transition zones as short as possible. In the case “Linear Distribution” is activated, the mesh transition zone will be in accordance with the settings specified on the mesh property.



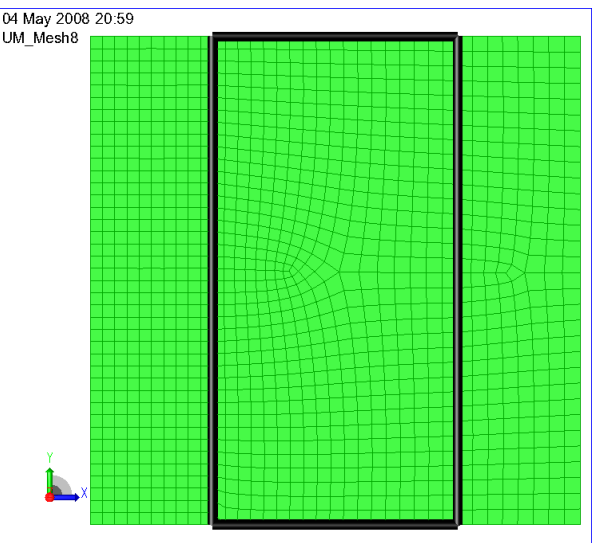
A model with 3 different mesh settings



The default mesh using Sesam quad mesher



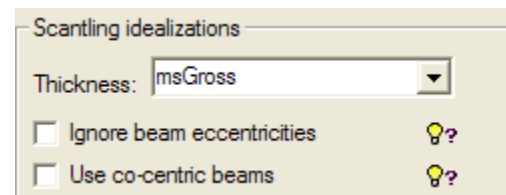
Sesam quad mesher
Linear distribution, growth rate 1.04
for Md_mid and Md_coarse



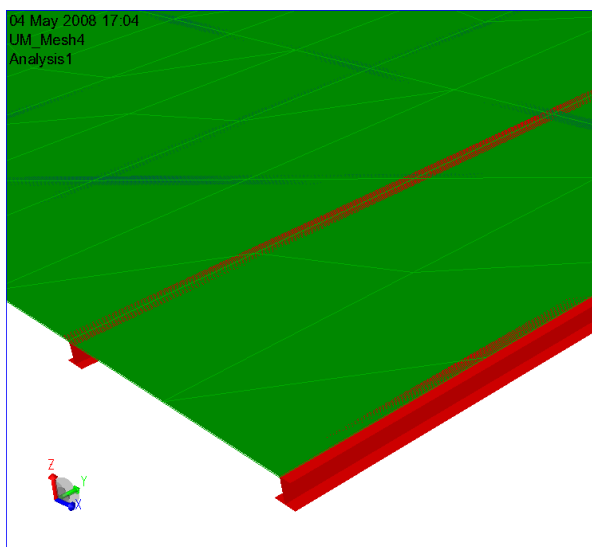
Sesam quad mesher
Linear distribution, growth rate 1.01
Internal edges for mid plate shown

6.2.1.11 Scantling idealizations

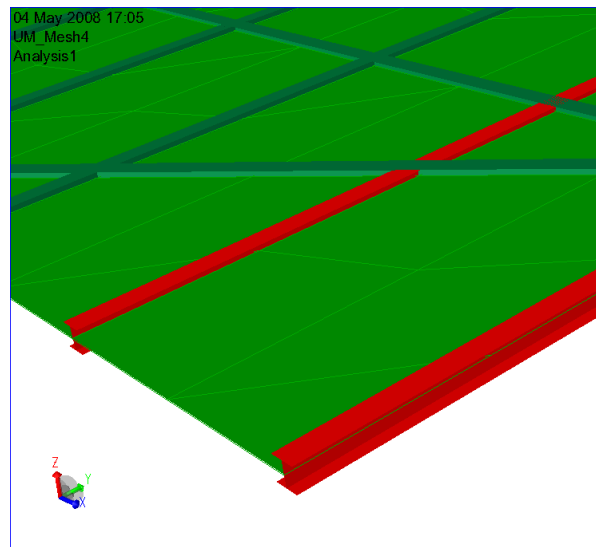
Usage and meaning of the “Thickness” combo box is described in chapter **Error! Reference source not found..** An example of usage is given in chapter **Error! Reference source not found.**



Beam and stiffener eccentricities in a finite element analysis are neglected by ticking off the option “Ignore Eccentricities”. In other words, the concept model may have eccentricities, but they are disregarded when making the finite element model. “Ignore beam eccentricities” should be checked when doing CSR bulk code check.

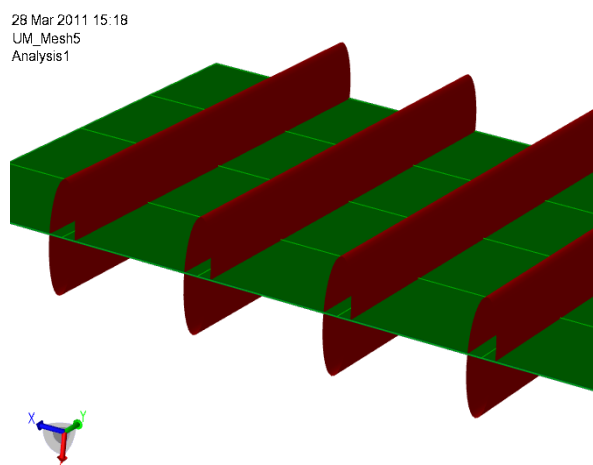


Finite element beams including eccentricities



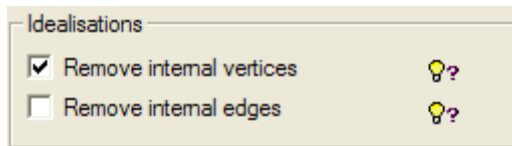
The eccentricities are disregarded

“Use co-centric beams” should be checked when doing CSR Tank code check. In this case an eccentric beam or stiffener will be converted to a general beam. Note that the beam must have an effective flange property for the option to have effect.



Finite element beams with effective flange property and “Use co-centric beams” checked.

6.2.1.12 Idealisations



When the "Remove internal vertices" option is checked (default) points not required to represent the model are eliminated. This option can be used to get better mesh around holes with buckling stiffeners.

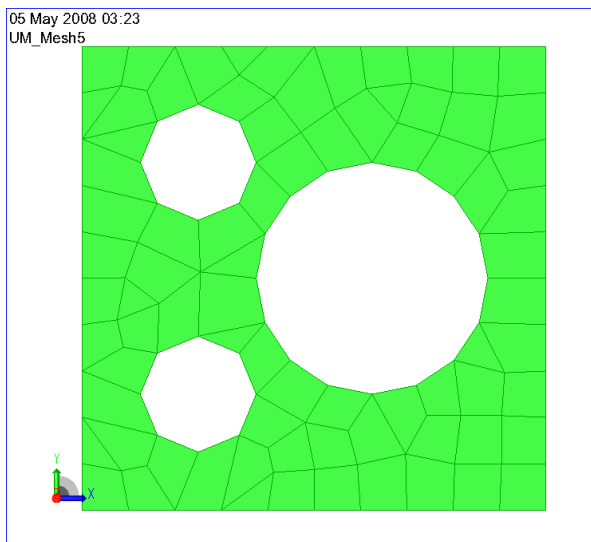
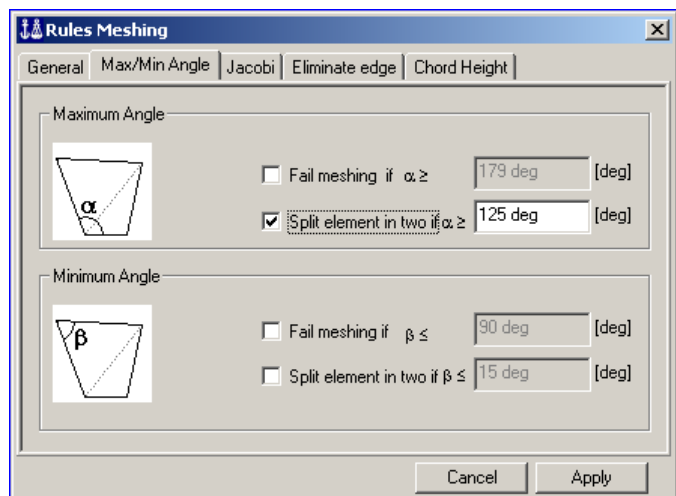
Edges not required to represent the model may be eliminated by checking the "Remove internal edges" check box. This option can be used to reduce the number of patches on a plate that needs to be meshed separately.

6.2.2 Mesh settings – max/min angles

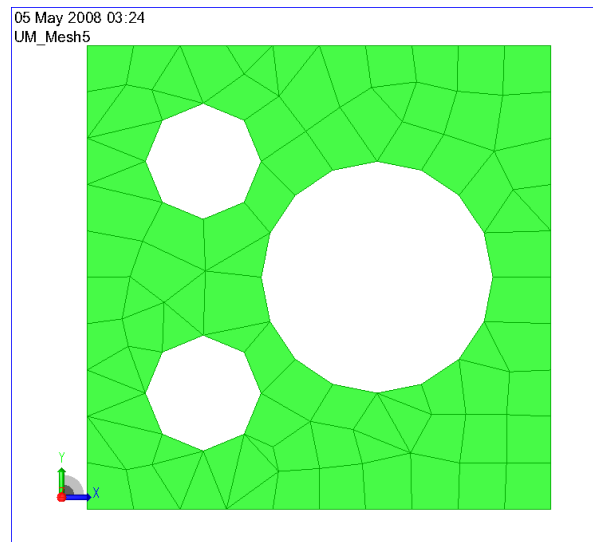
One of the philosophies behind the meshing is to provide a finite element mesh if possible. As a consequence the maximum allowable mesh angle is set to 179 degrees.

The quality of the mesh among others depends on the angles in a quadrilateral finite element. The *Tools/Analysis/Locate FE* will help you to document the angles used when creating the mesh, see previous Sections.

In the example below the meshing all quadrilateral elements with angles larger than 125 degrees are split in triangular elements.



Quadrilateral mesh with angles > 125 degrees



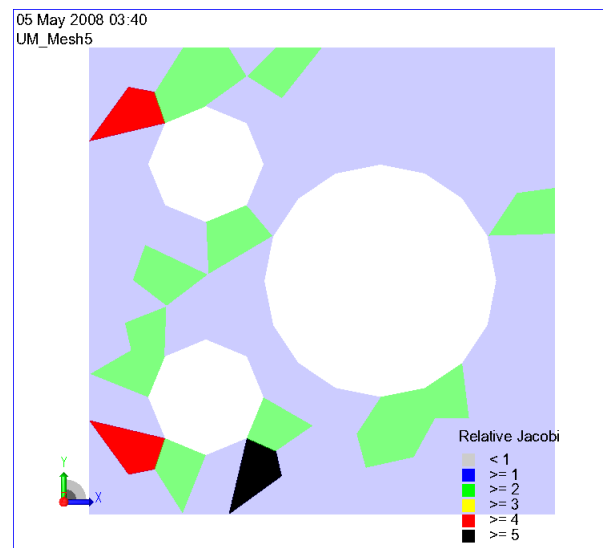
No quadrilateral mesh with angles > 125 degrees

6.2.3 Mesh settings – Jacobi

There are no limitations for GeniE to create a finite element mesh with respect to the relative Jacobi determinant. When running the finite element analysis (Sestra) warnings will be given for all relative Jacobi determinants higher than 4.0 – it is possible to de-activate this check in the analysis by editing the input file to be used by Sestra (see following Section).

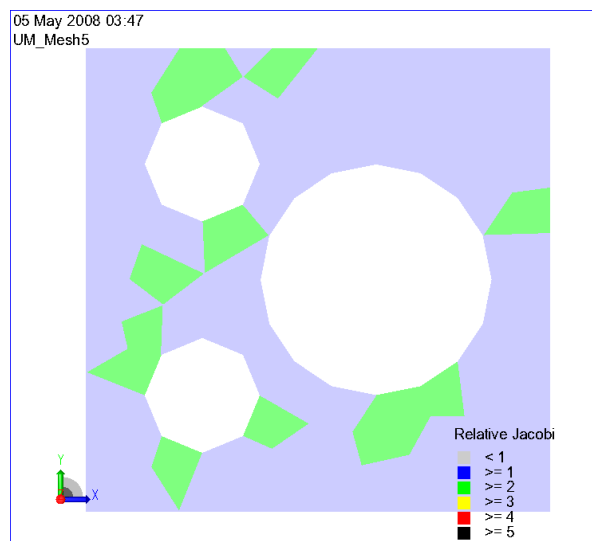
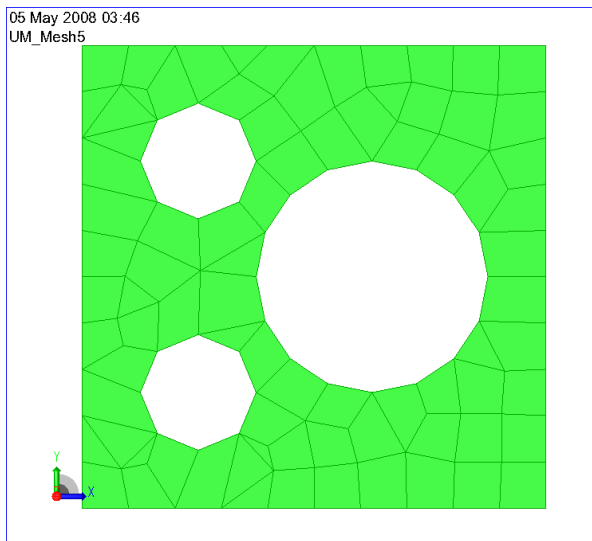
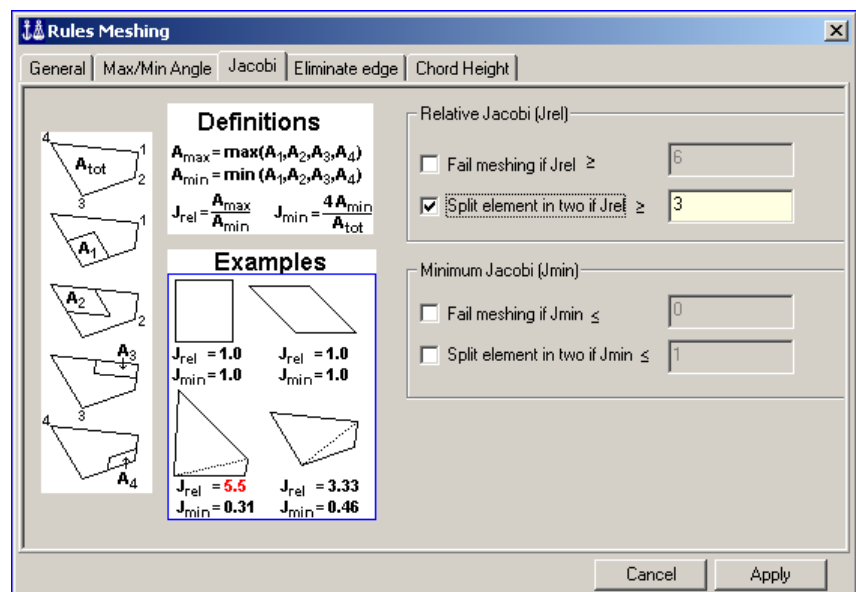
The Jacobi mesh rule allows you to control the relative Jacobi determinant either by stopping the meshing or splitting a quadrilateral in triangular elements if the determinant exceeds a threshold (maximum or minimum).

If the rule is set to fail when the relative Jacobi determinant exceeds 4.0 on the model above, no mesh will be created. The reason for this is that there is one plate only and the maximum value is above 4.0. If there are other plates in the model, these will receive a mesh if the Jacobi determinant test does not fail.



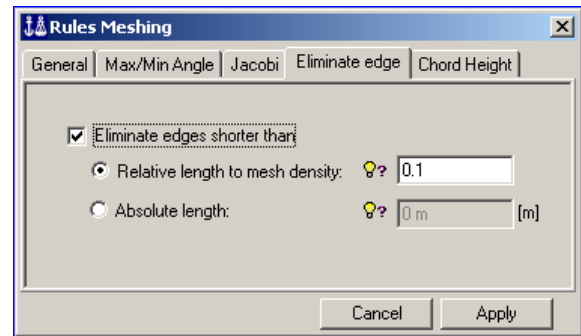
The example below shows that quadrilateral elements with a relative Jacobi determinant higher than 3.0 are split in triangular elements.

As can be seen, the highlighted elements above (red and black) are divided in triangular elements and there are no values higher than 3.0.



6.2.4 Mesh settings – eliminate edge

The “Eliminate Edge” option can be used to ignore changes in e.g. misaligned edges from typically weld seams and stiffener positions. The “Eliminate Edge” will perform an idealisation of the model, so this option should be used with care if the size of the edges to be eliminated is high. In addition to idealising the structure, this option may also effect the computation of finite element loads as compared to the conceptual loads.



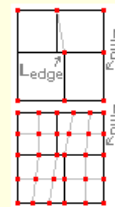
The option “Relative length to mesh density” should be used with care as this may lead to elimination of lengthy edges. However, for idealisation of small edges this option will in most cases give a satisfactory result.

Absolute Length

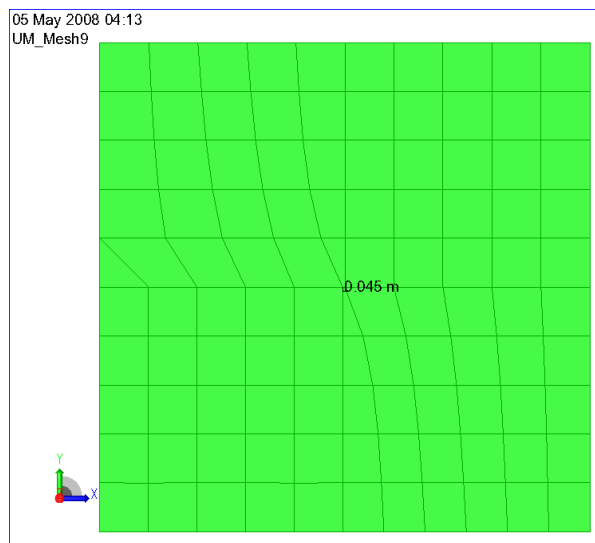
- Edges whose length is shorter than the specified value will be removed.

Relative Length

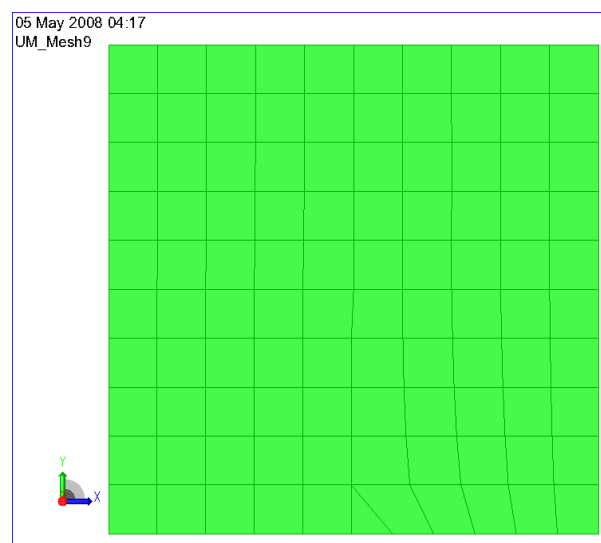
- Edges whose length relative to the mesh density is shorter than the specified value will be removed ($L_{edge}/L_{md} < \text{value}$).



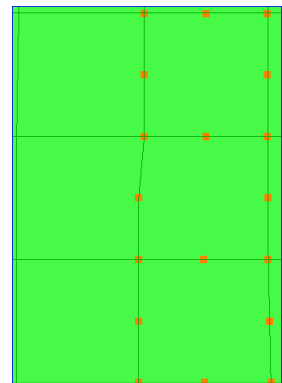
In the example below there is a misalignment of 45 mm between the weld seams in the middle of the plate.



No eliminate edge



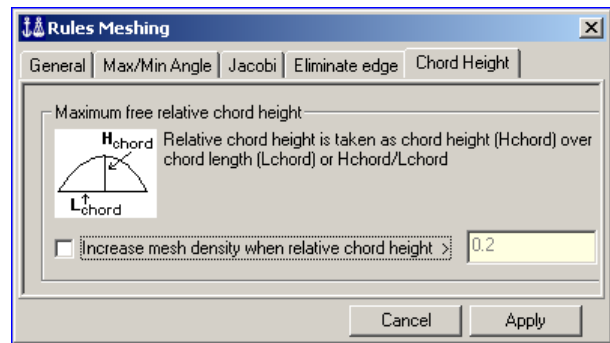
Eliminate edge using absolute value 0.05m
The idealisation is done over one finite element



6.2.5 Mesh settings – chord height

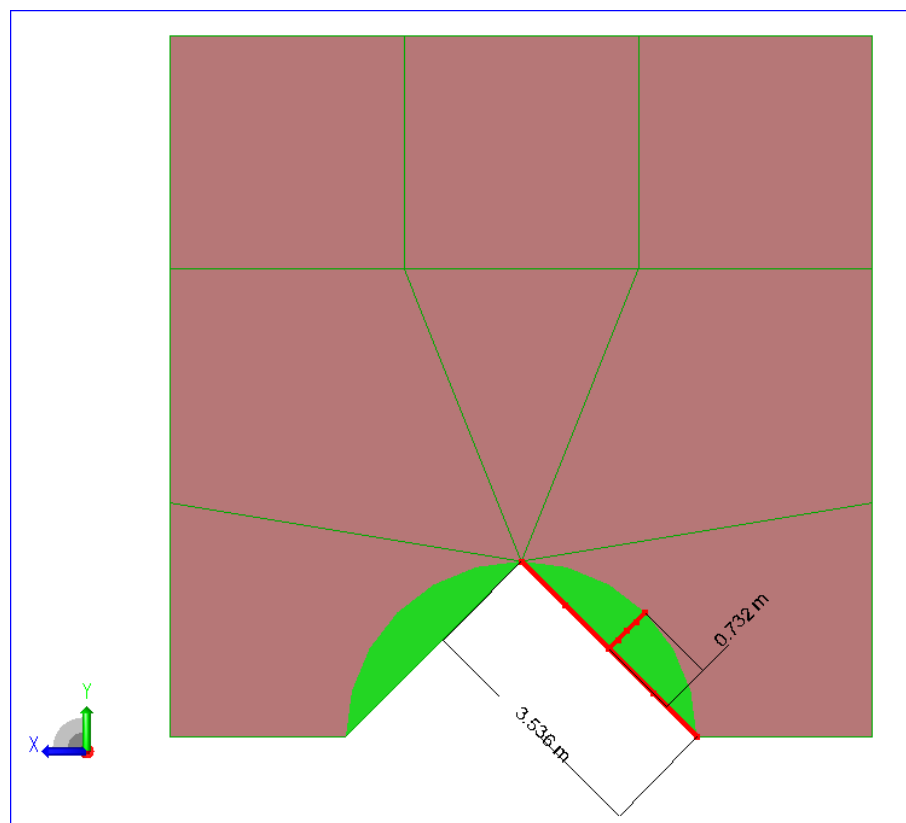
This option may be used for global models where you want to have some control of the mesh along curved details. It should be noted that the Advancing Front Mesher will give a better mesh and should thus be used for local models where stress results are important for such details.

The chord height option will increase the mesh density so that the minimum requirement for a relative chord height is satisfied. The relative chord height is the relation between the chord height and the chord length.

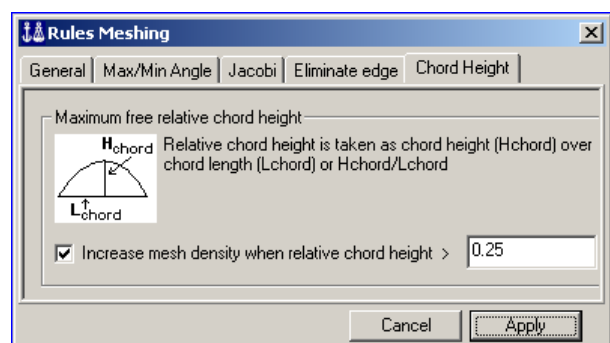


The functionality is shown in the following using a plate with a cut-out. The plate is meshed with a mesh density of 4 meters. The picture to the right shows the plate and the mesh (green colour) – one typical chord height (0.732m) and chord length (3.536m) are highlighted.

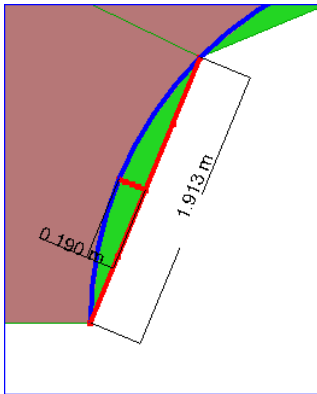
In this case the relative chord height is 0.207.



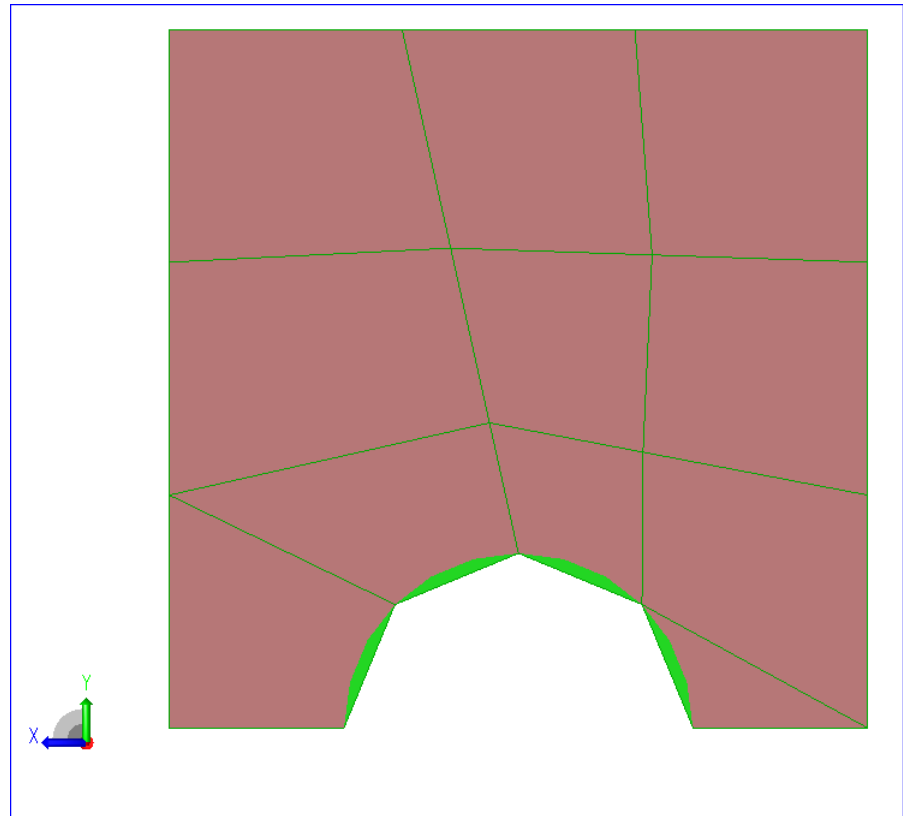
If the chord height option is used as shown to the right there will be no changes to the mesh layout. The reason for such is that the limit used (0.25) is larger than the actual relative chord height of 0.207 in the model above.



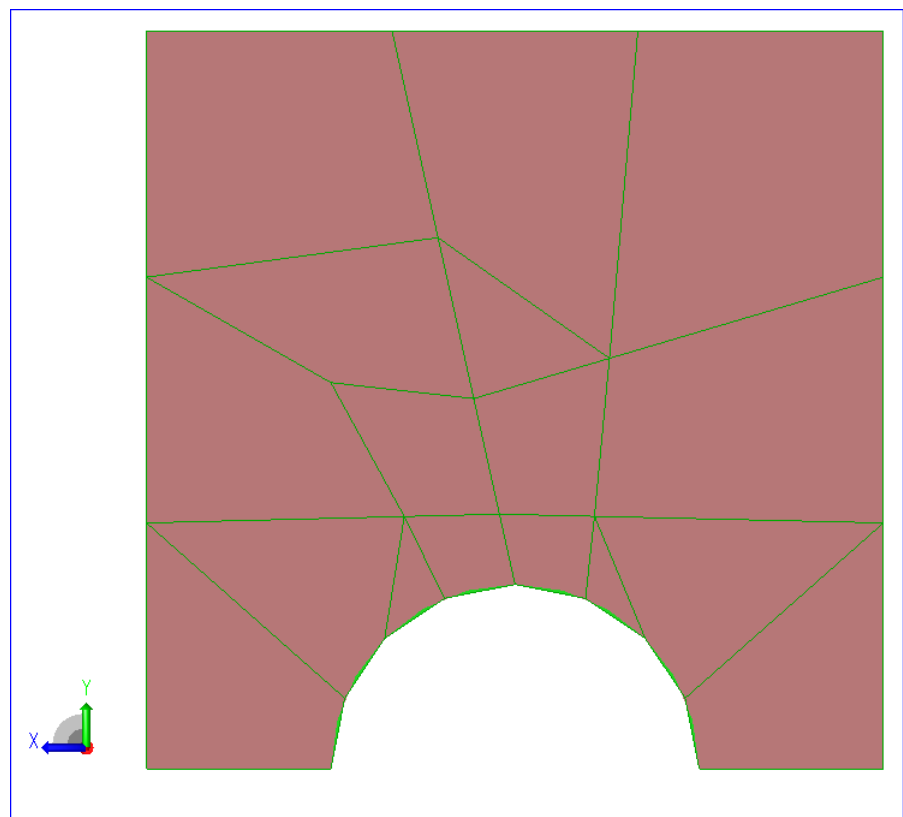
When the limit for the relative chord height is reduced to the half value ($0.207/2 = 0.1035$) the mesh is refined to satisfy the limit value.



As can be seen from above, a typical relative chord height becomes $0.190/1.913 = 0.1$ which is less than 0.1035.



A further refinement of the limit value for the relative chord height ($0.207/4$) leads to a further improvement of the mesh.



6.3 Local mesh settings

The global mesh settings may be over-rid by local mesh settings. Local mesh settings may be of types

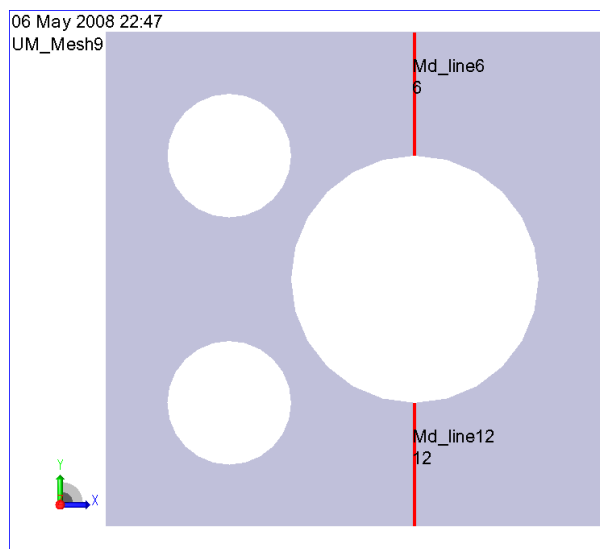
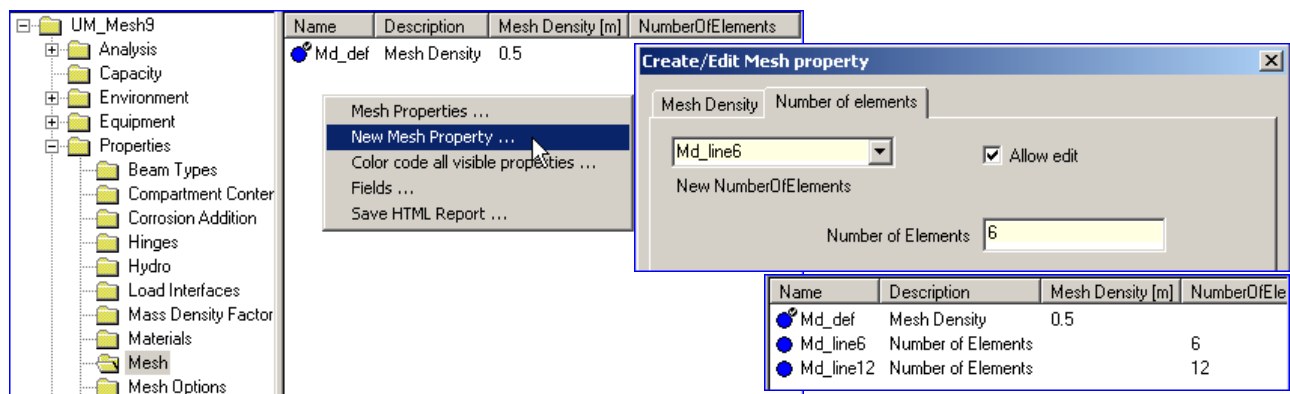
- Number of elements along a line or different mesh densities to various surfaces or lines
- Define specific feature edges (mesh control lines)
- Mesh options for face or edge
- Mesh locking

Each of these options is described in the following.

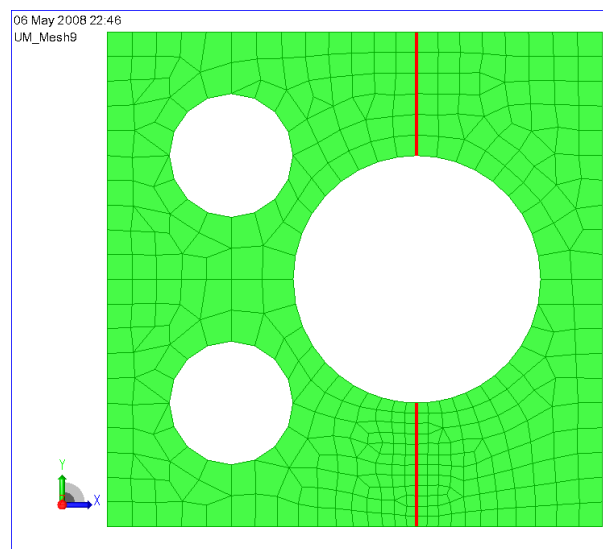
6.3.1 Number of elements along a line

The previous Sections described how to apply a general mesh density (or several) to plates, shells, beams and stiffeners. For beams it is also possible to define a specific number of elements (with equal length) along a line. A line can be a beam, stiffener, boundary curve or feature edge.

Number of elements along a line is defined from *Edit/Properties/Mesh Property/Number of elements* or from the browser as shown. In this case two properties are defined where number of elements are set to 6 and 12 respectively.



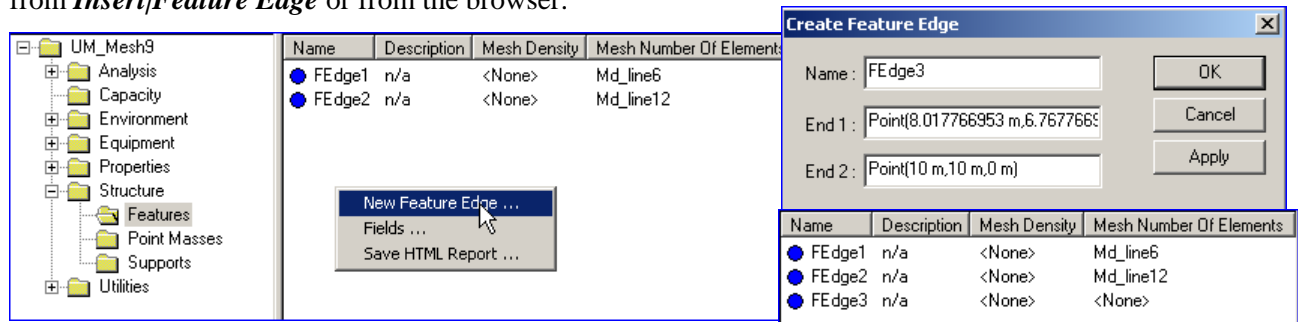
Number of elements assigned to lines



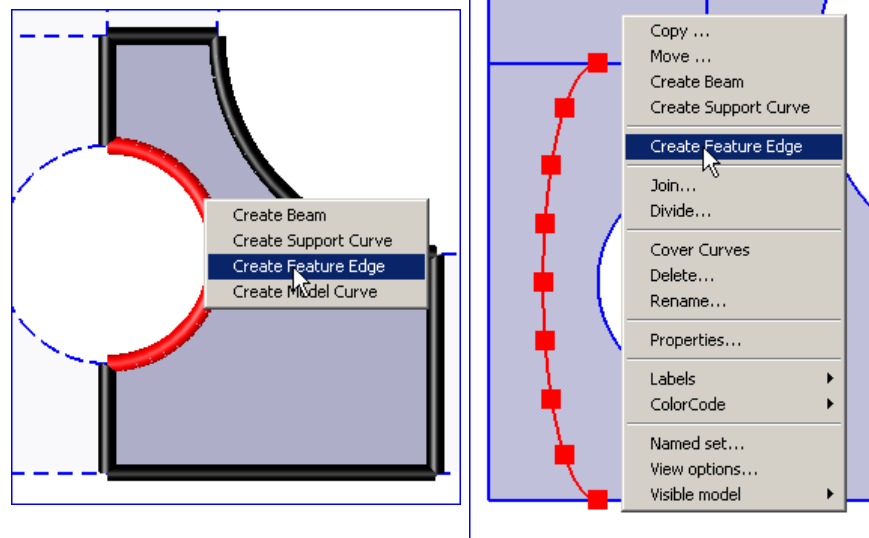
The new mesh with 6 and 12 elements along the respective lines. Advancing front quad mesher is used with element growth rate equal to 1.10.

6.3.2 Feature edges

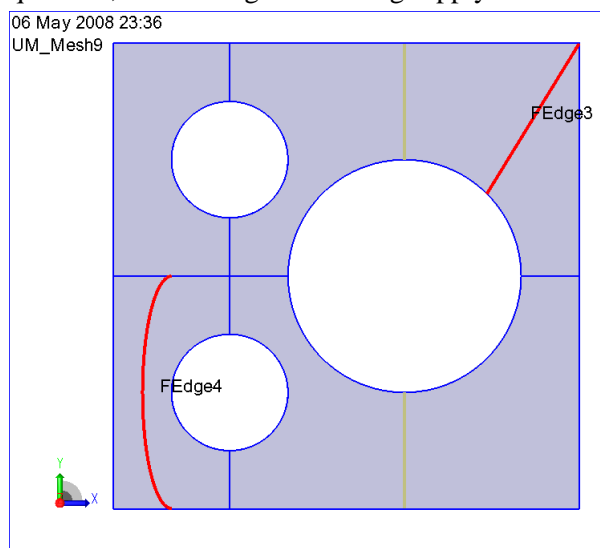
A feature edge is a line that can be inserted to control the mesh layout. It will define a topology edge on a surface between two points or along a curve. This means that the feature edge belongs to a plate or shell and it is not possible to insert a feature edge when there are no corresponding surfaces. Furthermore, when deleting a surface including a feature edge the feature edge will also be deleted. A feature edge is inserted from *Insert/Feature Edge* or from the browser.



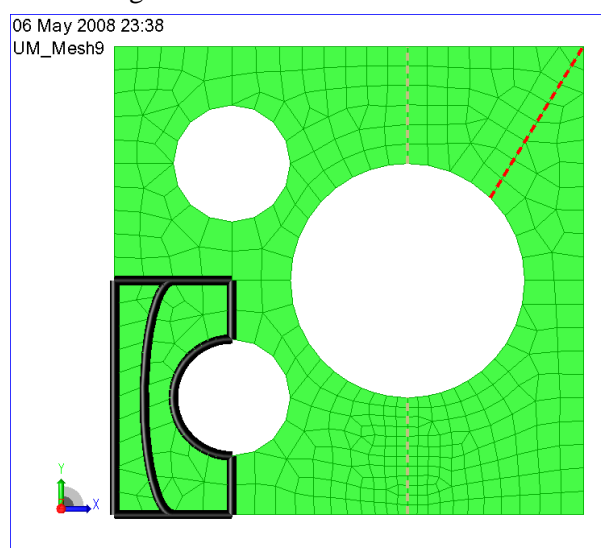
Feature edges may also be inserted by referring to existing lines or topology edges. Select the line or edge, **RMB** and click *Create Feature Edge*.



As mentioned in the previous Section, it is possible to add either a mesh density or a specific number of elements to a feature edge. In the following, there is no local mesh setting applied to the feature edges in question; hence the global settings apply for the new feature edges.



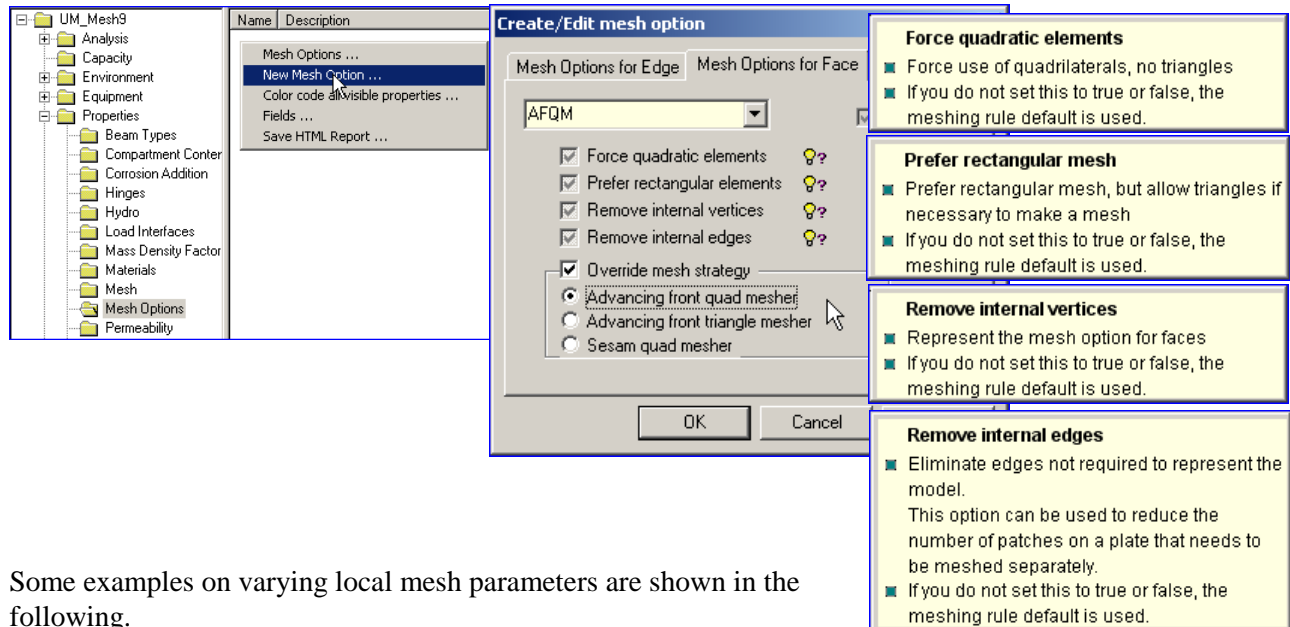
Feature edges *Fedge3* & *Fedge4* inserted



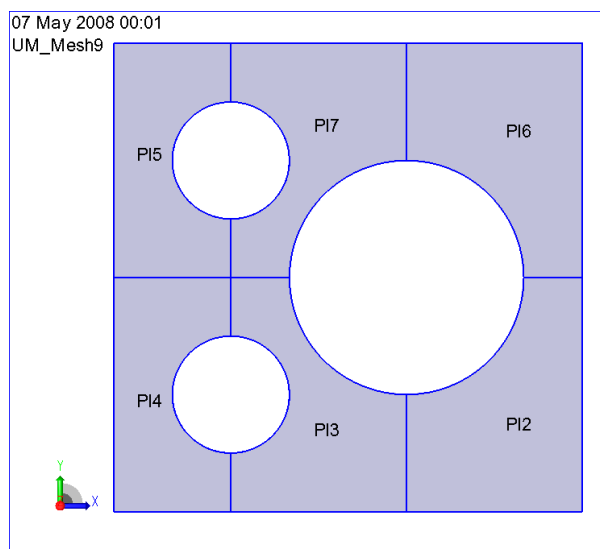
The feature edges defines topology edges

6.3.3 Mesh options for face or edge

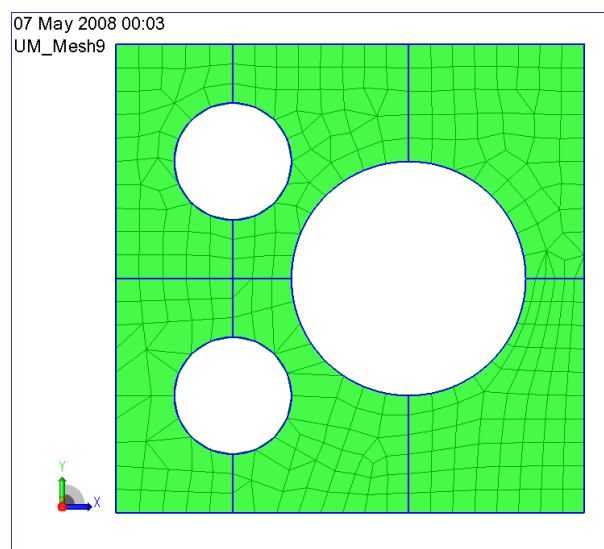
The global mesh parameters defined from **Edit|Rules|Meshing** may be replaced by local mesh parameters applied to specific parts of the model (surfaces, beams, boundary curves and feature edges). A local mesh parameter is defined from **Edit|Properties|Mesh Option** or from the browser. Please notice that local mesh options have no effect when the *Advancing Front Mesher* is selected (global or local setting).



Some examples on varying local mesh parameters are shown in the following.

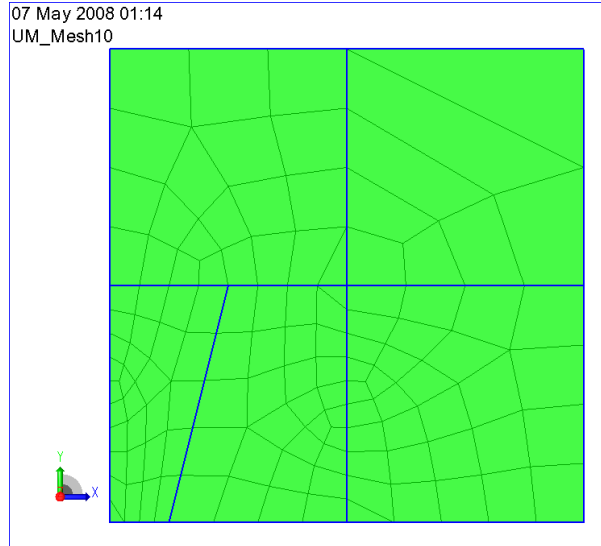
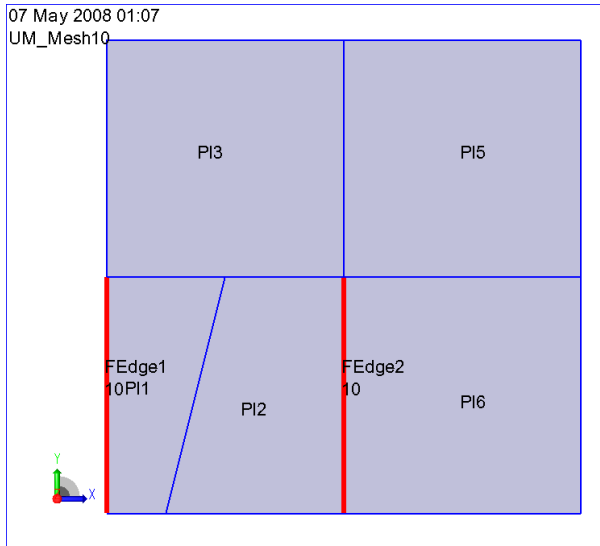


A model with 6 plates (PI2 -> PI7)

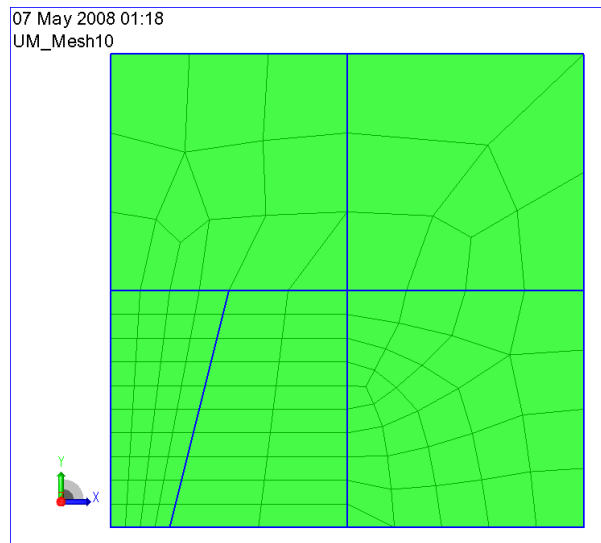
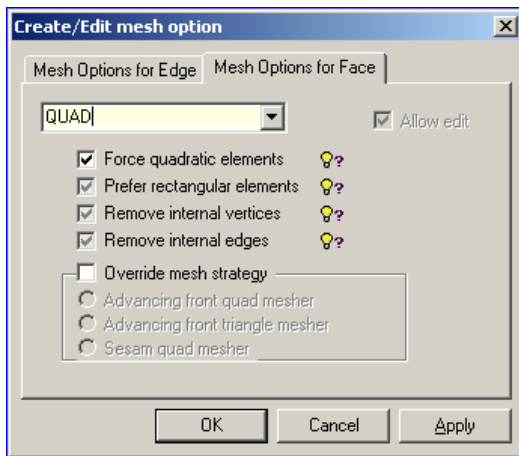


Local mesh parameters (above option AFQM) specifying *Advancing Front Quad Mesher* applied to plates PI5, PI6 and PI7. The rest of the plates are meshed with the global mesh setting (the *Sesam Quad Mesher*). Hence, the mesh is different on the upper part as compared to the lower part.

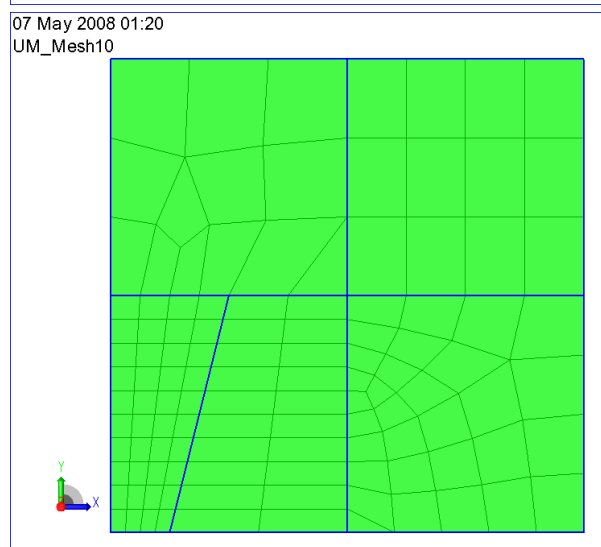
In addition to the global mesh default settings there are two local mesh controls in the model below. Each of the highlighted feature edges is assigned 10 finite elements along their lengths. The mesh based on these settings becomes highly irregular mainly due to the inclined connection between plates PI1 and PI2, but also because there is no global mesh density specified.



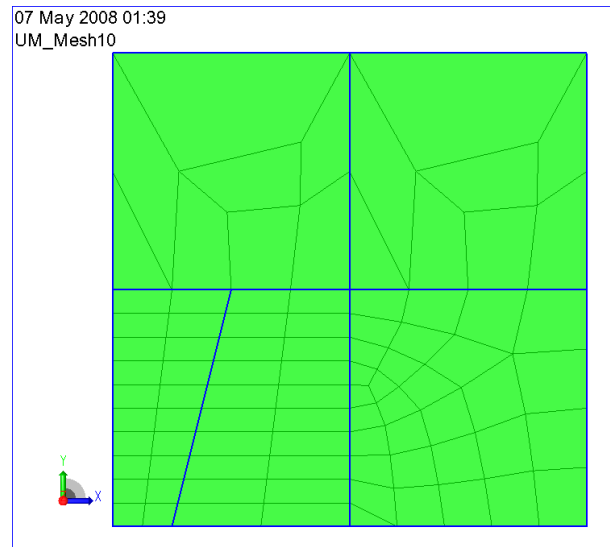
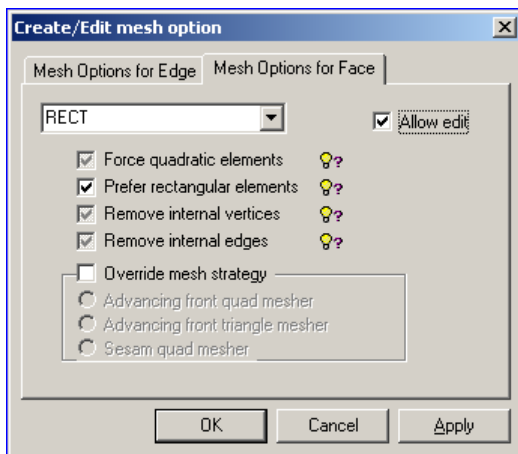
The mesh option *QUAD* is defined to force quadratic elements on plate PI2.



The mesh option *QUAD* is defined to force quadratic elements on plates PI2 and PI5.

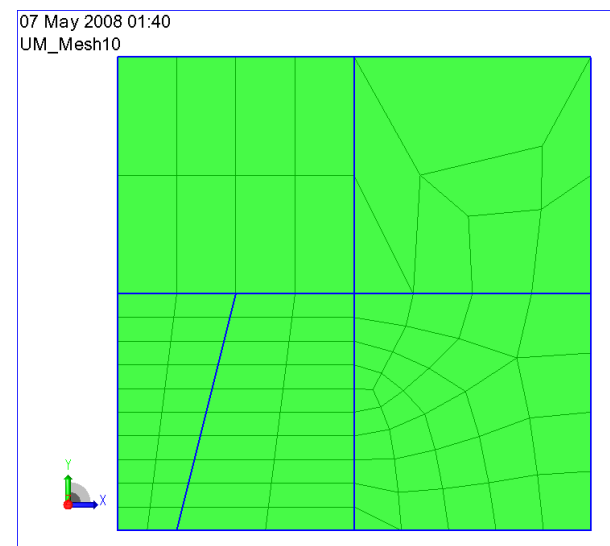


The mesh option *RECT* is defined to force quadratic elements on plates P11 and P12.



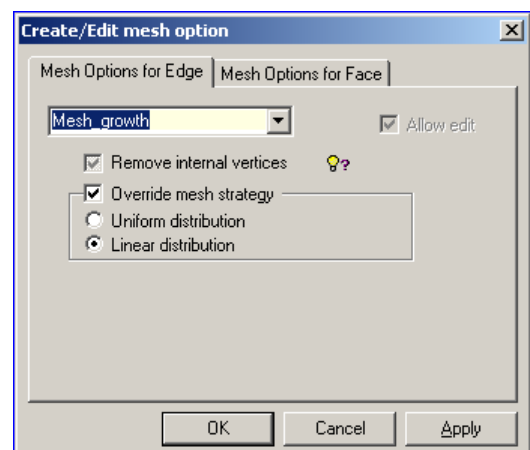
The mesh option *RECT* is defined to force quadratic elements on plates P11, P12 and P13.

As can be seen, rectangular elements are now made as compared to a more quadratic shape as in the previous example when using the *QUAD* property.



For the two other options Remove Internal Edges and Remove Internal Vertices, please see the definitions in Section *Global Mesh Settings* for further details.

The parameter settings for Mesh Options for Edge works the same way as for Mesh Options for Face, but there are fewer options. For a description on how to use a linear distribution of the mesh in transition zones, please see previous Section *Global Mesh Settings*.



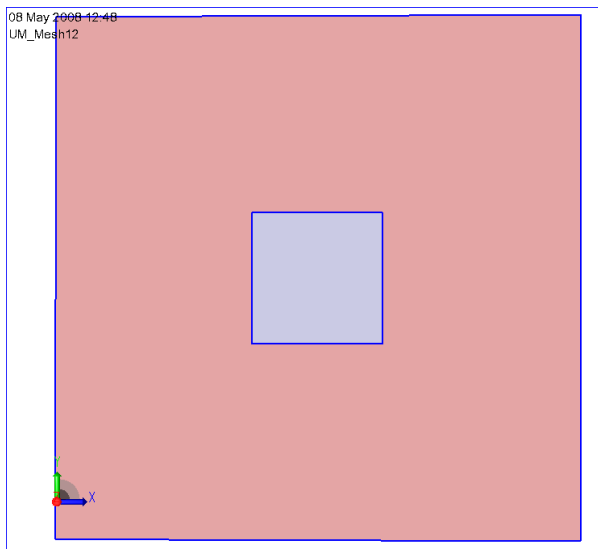
6.4 Refine mesh zones

GeniE will automatically create the mesh in the transition between surfaces with different mesh densities. By using the mesh growth parameter as shown in the previous Section it is possible to control the length of the transition zone (or how many elements to be used from e.g. coarse to fine mesh).

It is advised that you use regular edges (by defining surfaces, beams or feature edges) when changing from a mesh density to another. This will guide the program to make a best possible mesh in the transition part. The use of *Advancing Front Mesher* in these regions will in most cases improve the mesh quality, but this depends on where you want the best quality; in the middle of a surface (*Sesam Quad Mesher*) or along its edges (*Advancing Front Mesher*).

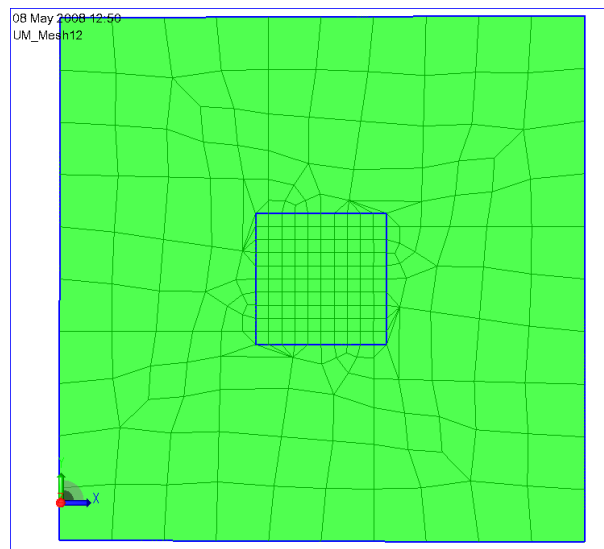
For high control of the mesh you may use number of elements along a line (typically edge of a surface) to achieve a very regular finite element mesh.

Some examples are shown in the following.



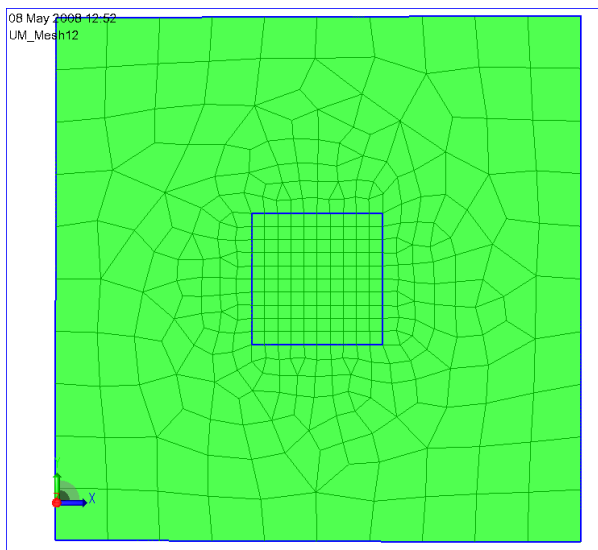
Regular plates

Mesh densities: Inner 0.25m, Outer 1.0m

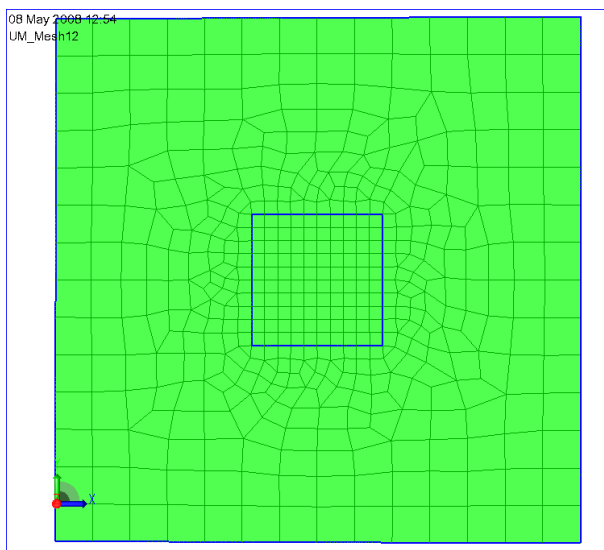


Sesam Quad Mesher

Program default settings



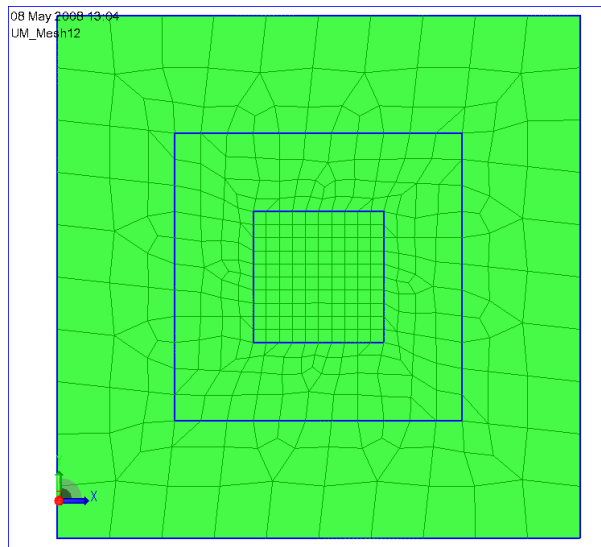
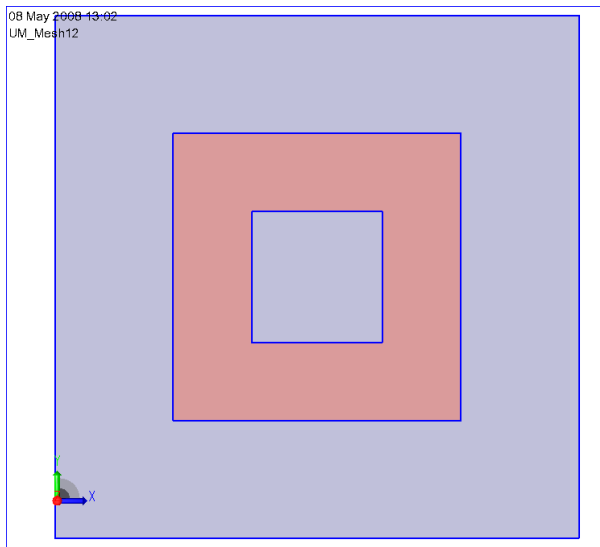
Advancing Front Mesher



Advancing Front Mesher

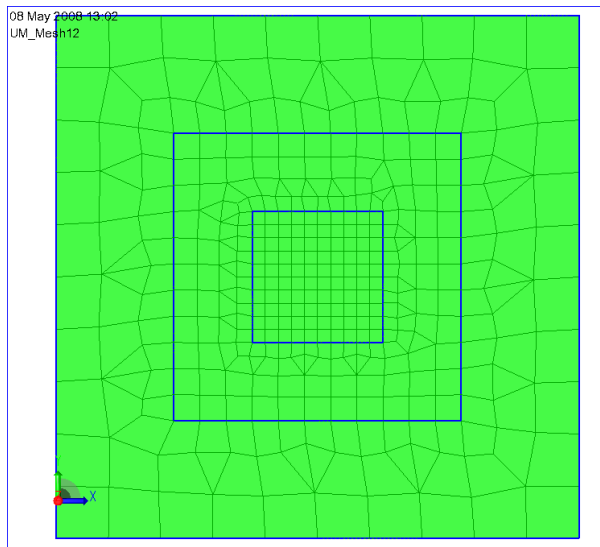
Growth rate 1.10 on outer plate

The mesh can be improved by inserting an extra rectangular zone (in this case by adding a new plate). New edges are now part of the model and this will guide the meshing in addition to the other settings.



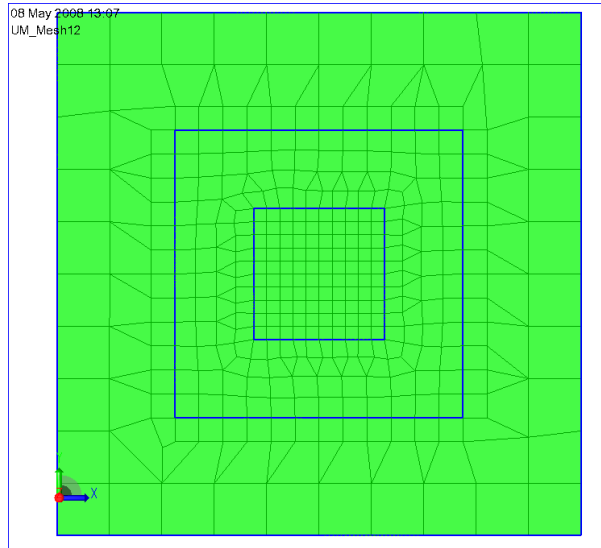
Regular plates

Mesh densities: Inner 0.25m, Mid 0.5m, Outer 1.0m



Sesam Quad Mesher

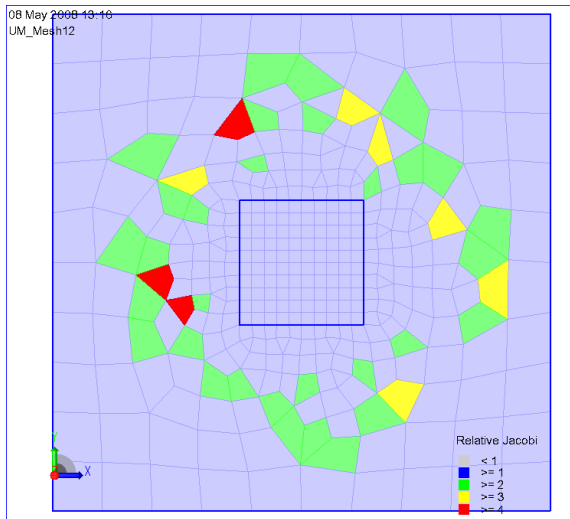
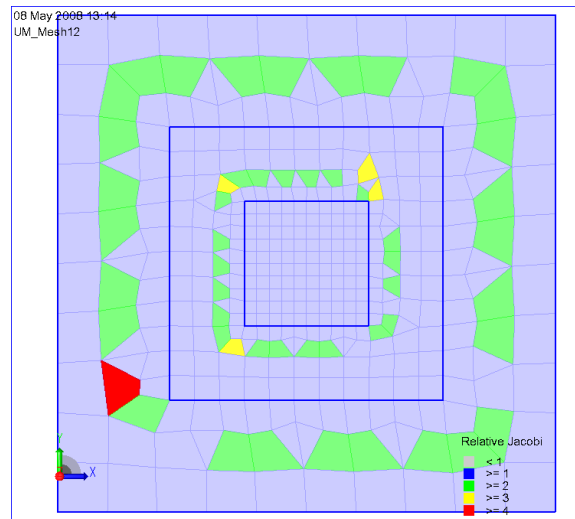
Program default settings



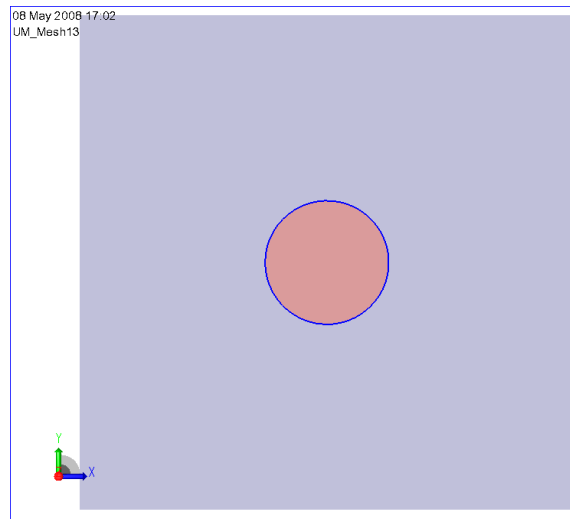
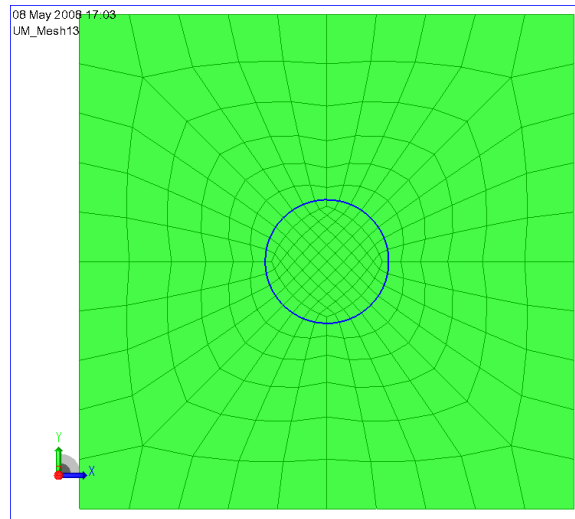
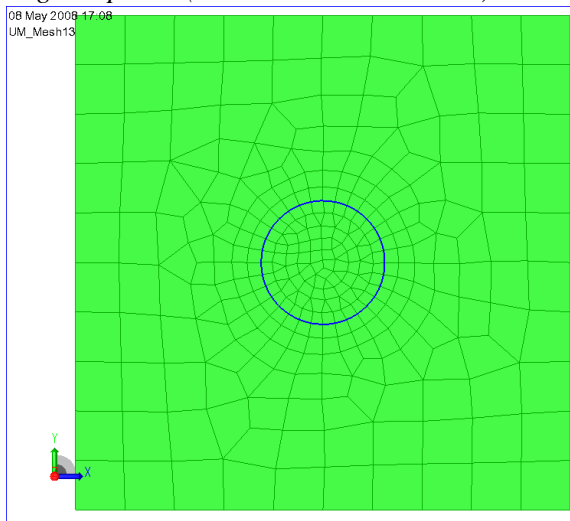
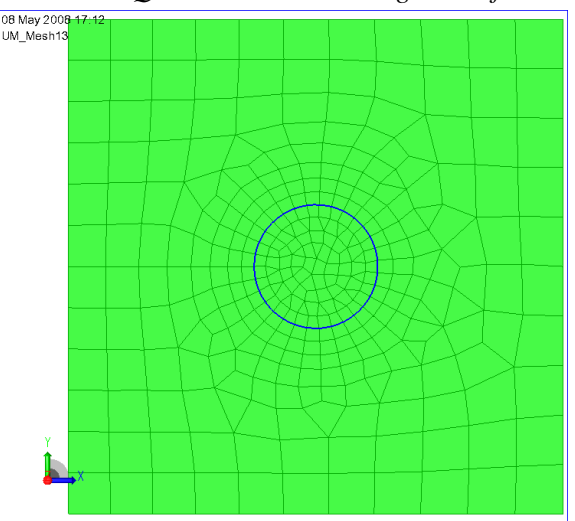
Advancing Front Mesher

*Advancing Front Mesher
Growth rate 1.10 on mid plate*

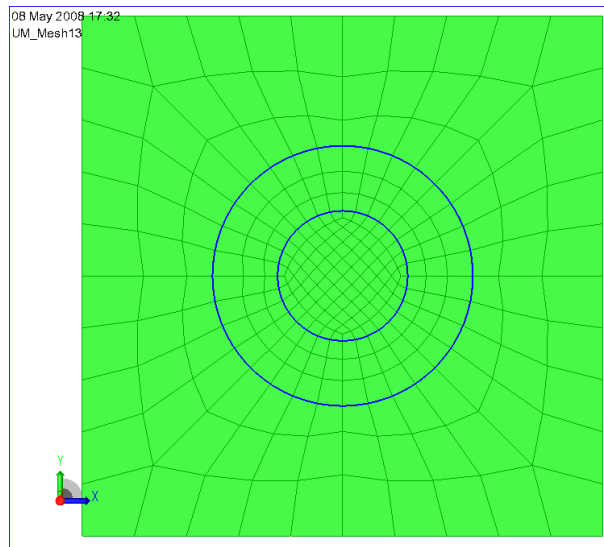
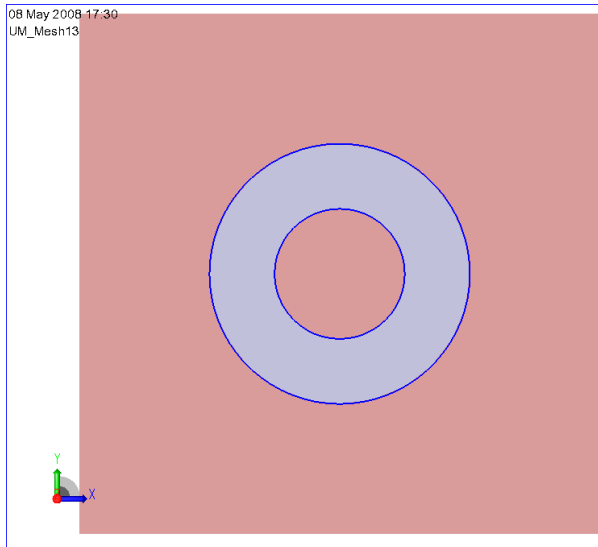
As can be seen from the example above, the introduction of an extra mesh transition zone significantly improves mesh quality when using the Sesam Quad Mesher. The mesh created when using the Advancing Front Mesher has relatively high quality for the plates without the extra mesh transition zone. However, an extra layer will increase the mesh for this mesh option too.

*Relative Jacobi determinants with no extra zone**Relative Jacobi determinants with extra zone*

The following example shows a plate with a circular part in the middle.

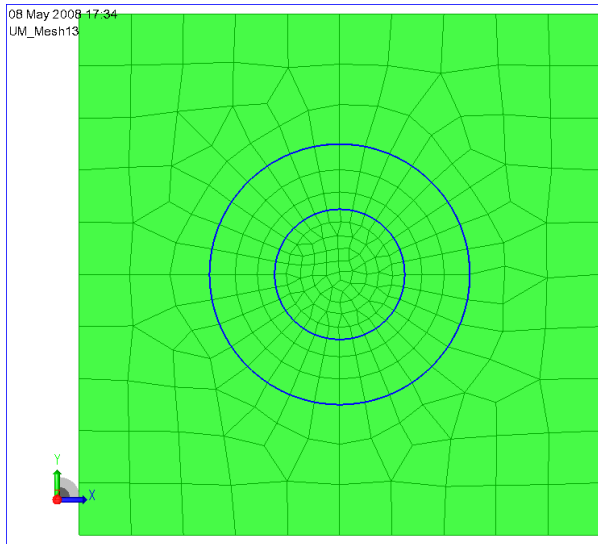
*Regular plates (Inner 0.25m Outer 1.0m)**Sesam Quad Mesher and Program default settings**Advancing Front Mesher**Advancing Front Mesher & Growth rate 1.1*

The mesh can be improved by inserting additional mesh transition zones (in this case one zone).

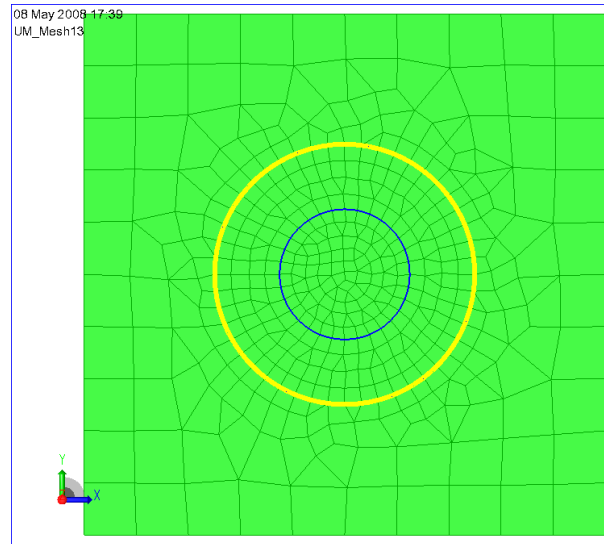


Regular plates (Inner 0.25m, mid 0.5m Outer 1.0m)

Sesam Quad Mesher and Program default settings



Advancing Front Mesher



Advancing Front Mesher & number of elements along circles: Inner 32, Outer 48

As can be seen the mesh created by the Sesam Quad Mesher improves significantly by inserting an extra mesh transition zone. This also applies to the Advancing Front Mesher, but this may not be so necessary to do since the mesh has a good initial quality.

6.5 Prioritized meshing

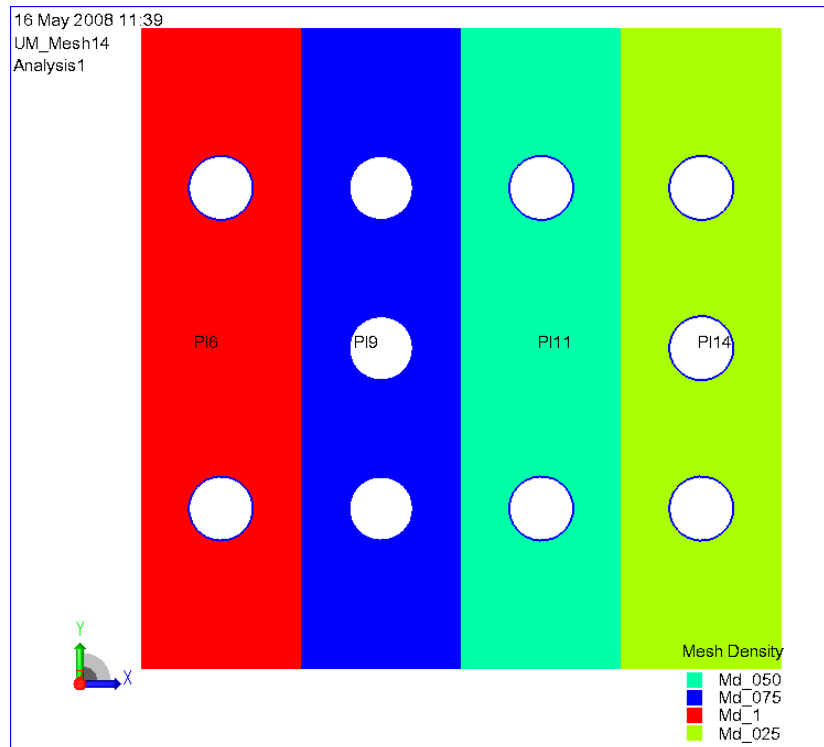
When GeniE is making a mesh it randomly picks the starting mesh position and the meshing sequence. However, when the structure is not modified the mesh will be the same every time a mesh is made.

It is possible to specify the mesh sequence to control the meshing sequence. Typical examples where this technique can be used is when you want the best mesh to be created for critical parts. To do this you need to add mesh rules to the analysis activity. To make an analysis activity is described in the next Section.

The effect of prioritized meshing is shown using the example to the right.

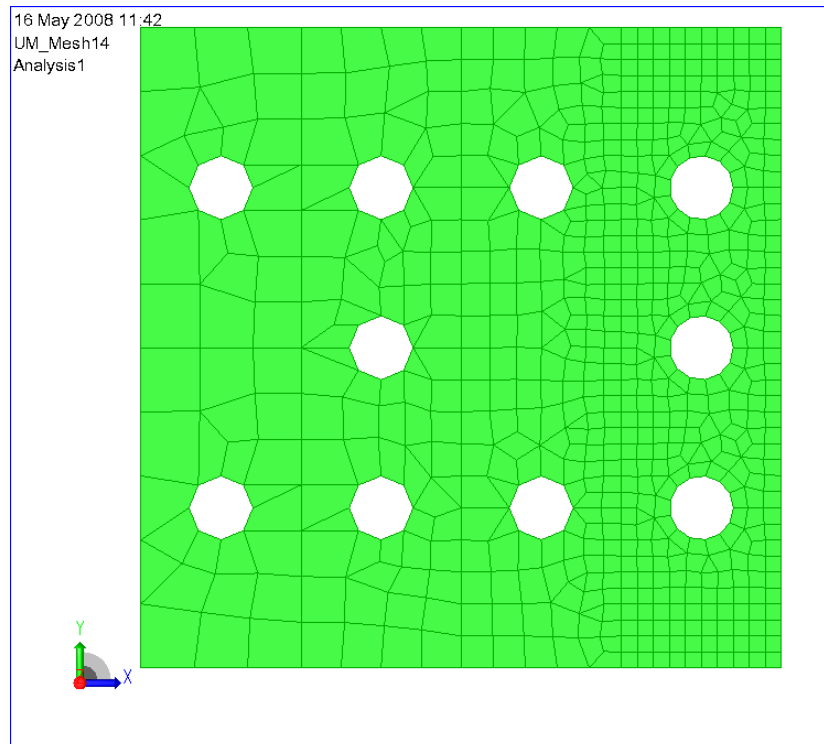
There are four plates each with different mesh density.

The meshing algorithm is Advanced Front Mesher; the remaining mesh parameters are system defaults.



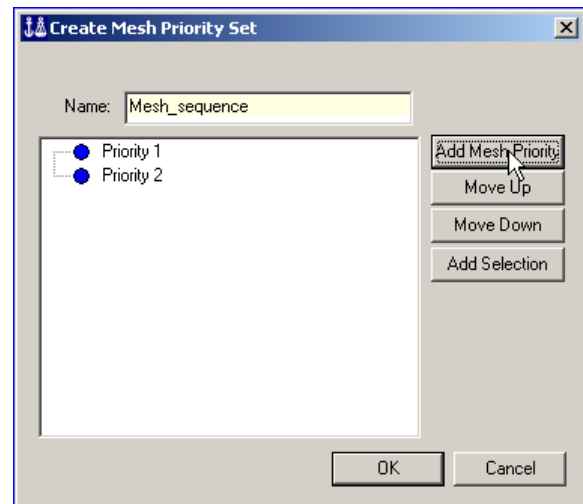
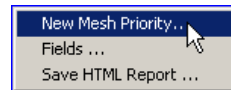
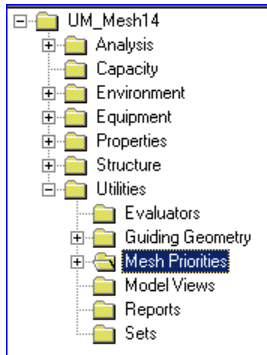
The mesh using above settings looks like the layout shown to the right.

The mesh has a steady growth from coarse to fine.



To control the meshing sequence you can use prioritized meshing. You may specify several layers of priorities and each priority may contain several objects (plates and beams). The properties for prioritized meshing are connected to the meshing activity. This means that it is possible to generate different mesh for the same concept model.

The property for prioritized meshing is defined from the browser area, Right click the folder Mesh Priorities and select New Mesh Priority. Then define number of priority levels (you may modify number of levels and their content later). In the example below two priorities are defined.



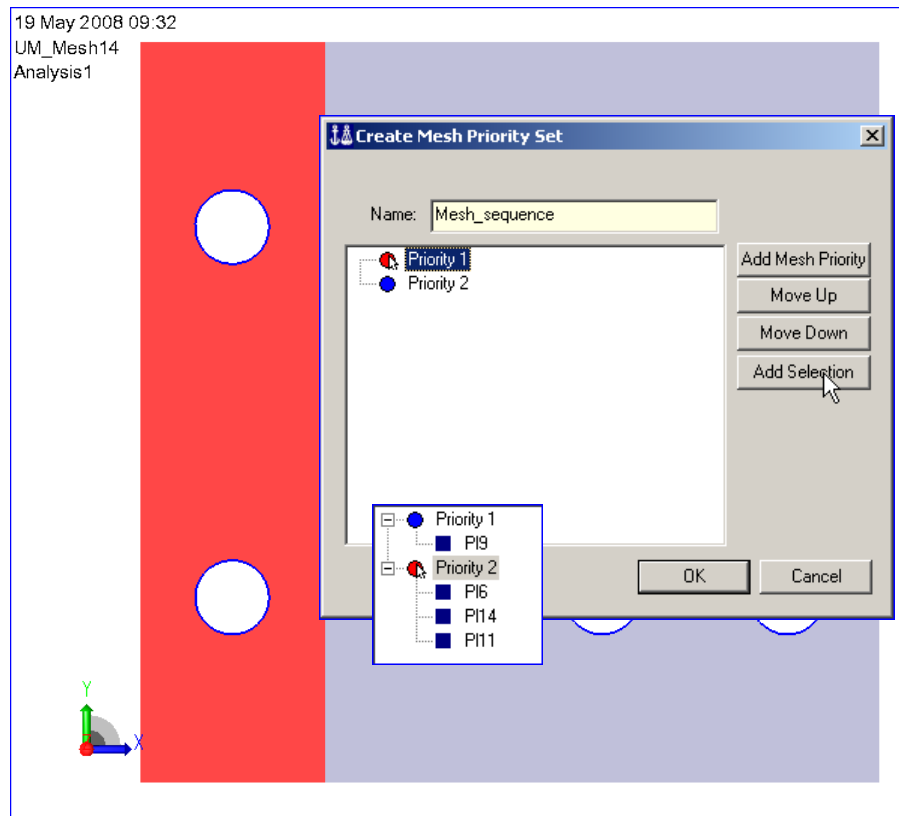
To add a plate or beam to a mesh priority:

1. Select the object(s)
2. Select the priority
3. Click "Add Selection"

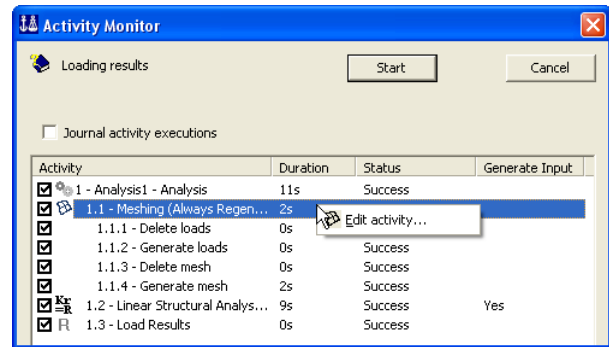
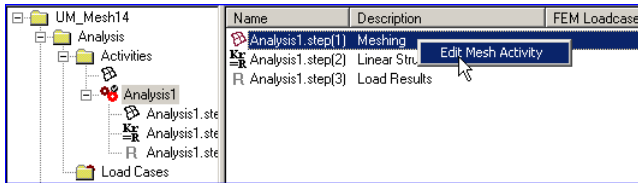
The example to the right shows that two levels have been defined and that the plate P19 is part of the first priority.

The rest of the plates are part of the second meshing priority.

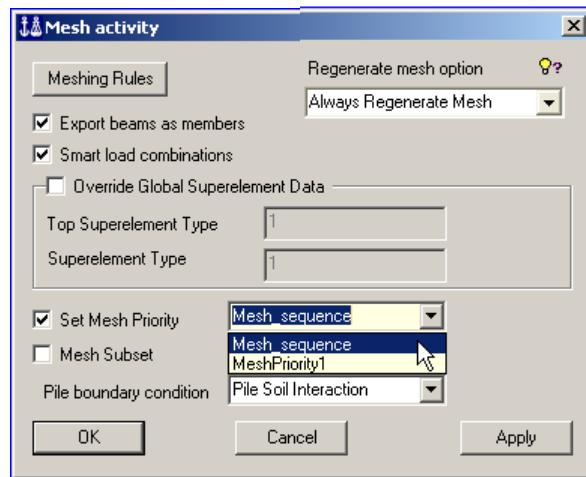
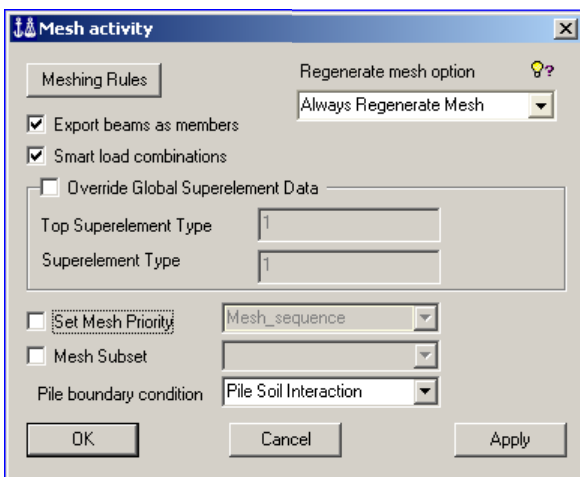
Notice that any plates or beams not part of a meshing priority level will be meshed at the end.



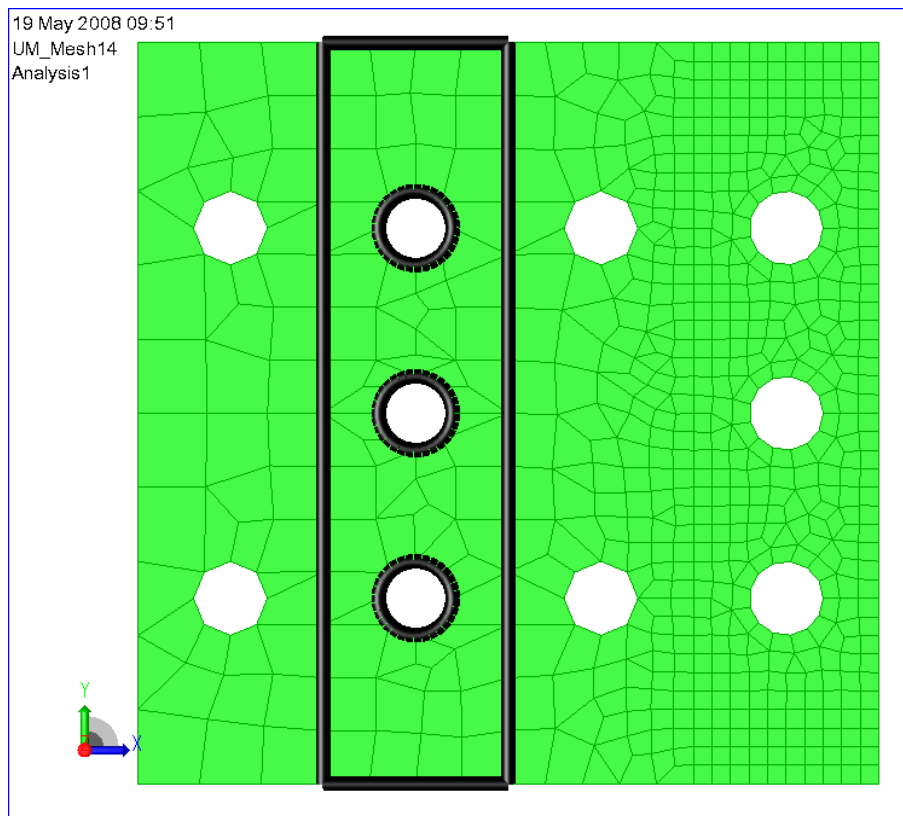
Mesh priorities are included in the meshing from the browser or the activity monitor (see the following Section for details on analysis folder and activity monitor).



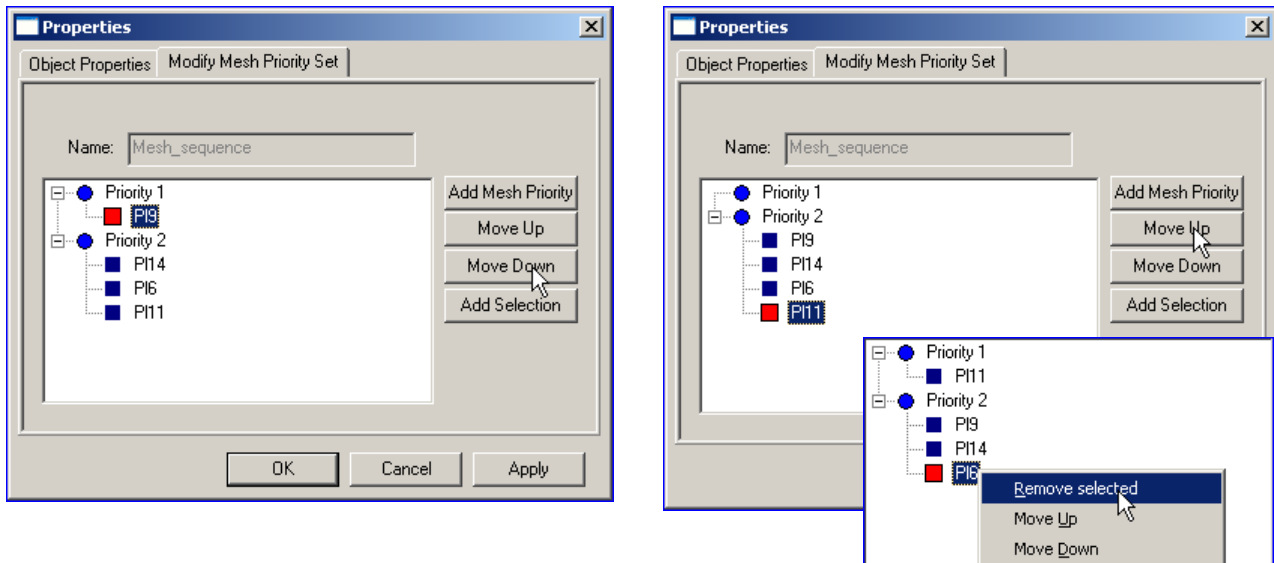
The pictures below show the default set up for meshing and how to include mesh priorities. Notice that you also can access the meshing rules from here. The default mesh is shown on the previous pages while the mesh using the mesh priority “Mesh_sequence” is depicted below.



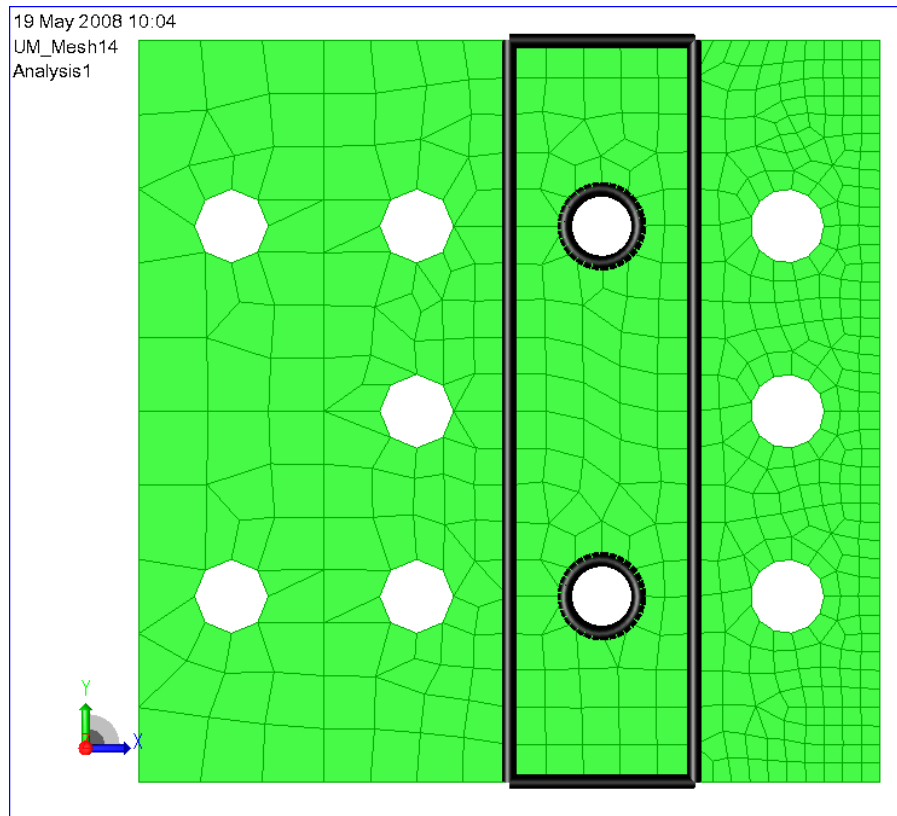
In this case the plate P19 is meshed before the other plates. The mesh from P19 now acts as starting conditions when meshing the remaining plates.



The example below shows that P19 is moved to level 2 while P111 is moved to level 1. Furthermore, plate P16 is removed from the prioritised meshing; i.e. it is meshed after the plates that are part of the prioritized meshing.



The new mesh will look like:



6.6 Mesh parts of structure only

It is possible to mesh parts of the structure and make different analysis models from one and the same concept model. This technique can be used to for example define different finite element models and use them to build a superelement assembly or to create local analysis models.

Meshing parts of structure is often related to running analysis, and more details may be found in the following Chapter.

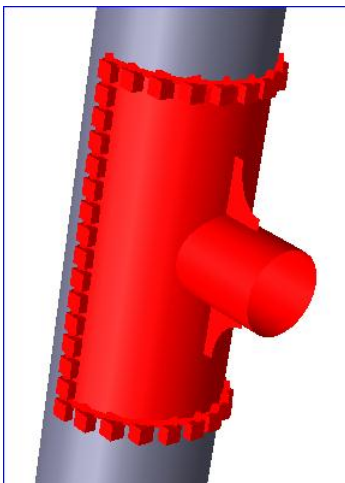
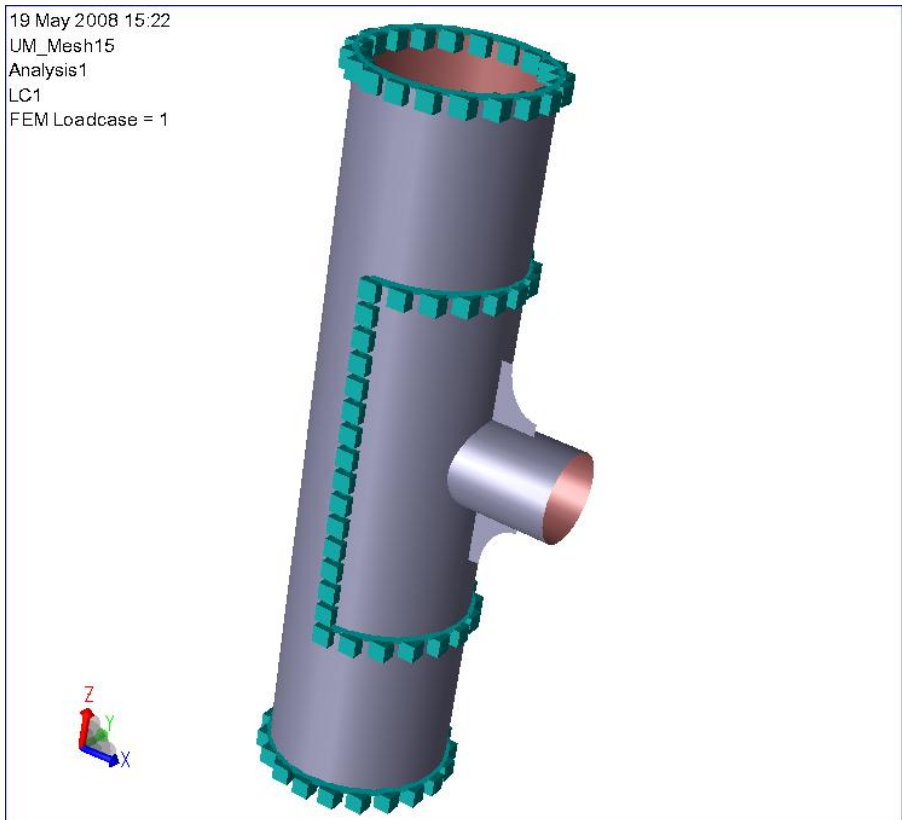
The procedure to do this is to make a set that also can include boundary conditions and limit the meshing to such. All loadcases will be included, but only with loads that are relevant for the active selection. Typically, the self weight load case will contain effect from the selected structure only.

A tube intersected with a brace is used to show how to create different finite elements for parts of the same concept model.

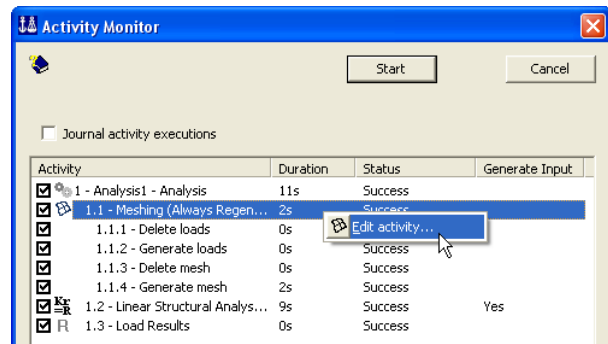
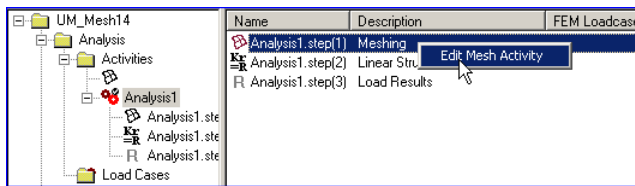
The model includes boundary conditions for both the global model (the whole structure) and for the edges describing a local model.

Two sets are made; the set *Global_model* includes everything except the internal boundary conditions. Similarly, the set *Local_model* describes the part close to the incoming brace including the internal boundary conditions (see picture below).

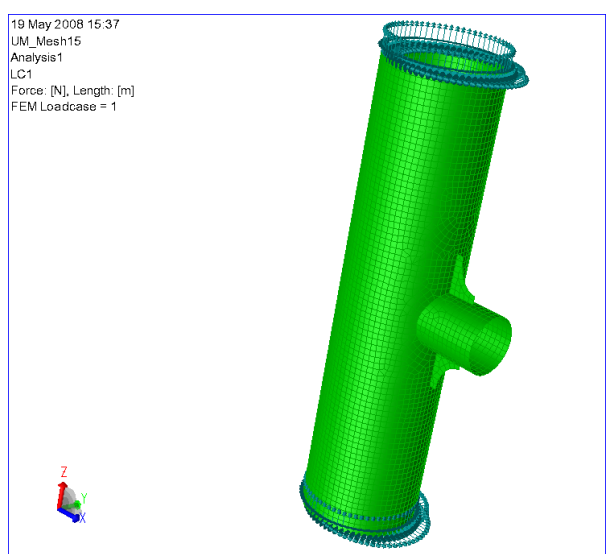
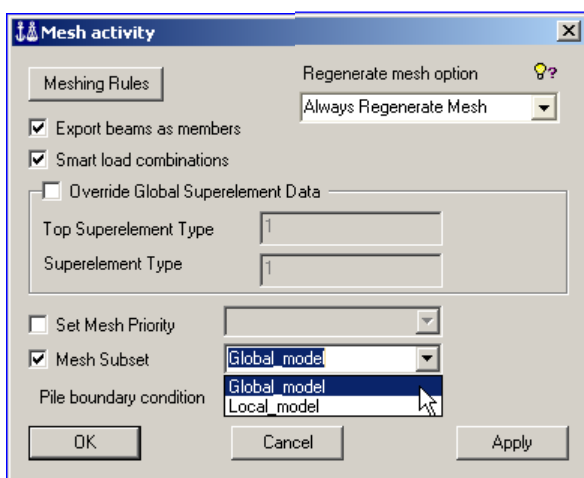
19 May 2008 15:22
UM_Mesh15
Analysis1
LC1
FEM Loadcase = 1



Per default GeniE will mesh the entire structure. To specify which parts (or which named set) to be meshed you need to edit the mesh activity from the browser or from the activity monitor.

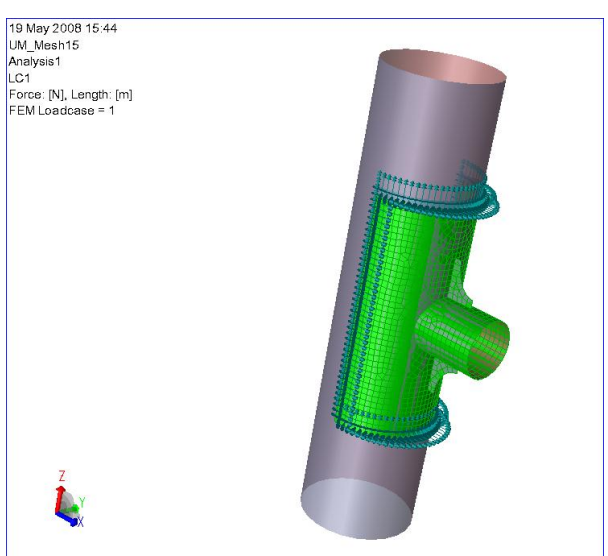
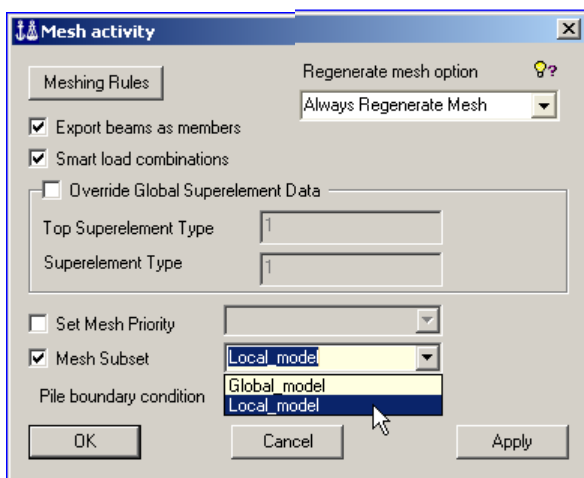


In the first example the named set *Global_model* is used when making the finite element model. The corresponding mesh is shown.



Notice that the boundary conditions that are not part of the named set *Global_model* are not included in the finite element model.

When making a finite element model based on the named set *Local_model*, the mesh includes only the detailed part. It is also possible to change the mesh density between such models – this is a technique often used to make refined analysis models.



The picture to the right also shows the parts that are not included in the finite element model.

6.7 Mesh locking

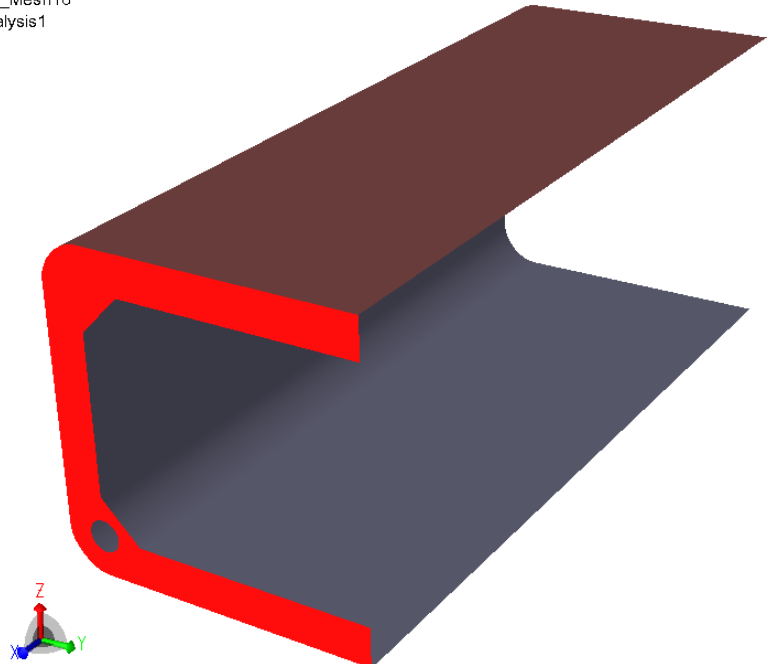
Mesh locking ensures that parts of the mesh will remain the same when you have declared a mesh satisfactory by locking its mesh coordinates. Typical scenarios are when you model 2D parts (like a web-frame) and assemble these into a 3D structure (like a hull with web-frames).

Mesh locking is often used together with mesh parts of the structure where the mesh coordinates are locked before they are used in a global model. Also notice that the mesh locking is a property which means that when you copy a structural member the mesh lock co-ordinates are also part of the copy operation.

A small example on how to use this methodology is given in the following.

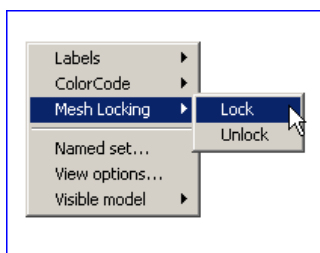
The goal is to tune the mesh for the web-frame and ensure that the mesh is reproduced for all instances of the same web-frame.

19 May 2008 16:13
UM_Mesh16
Analysis1



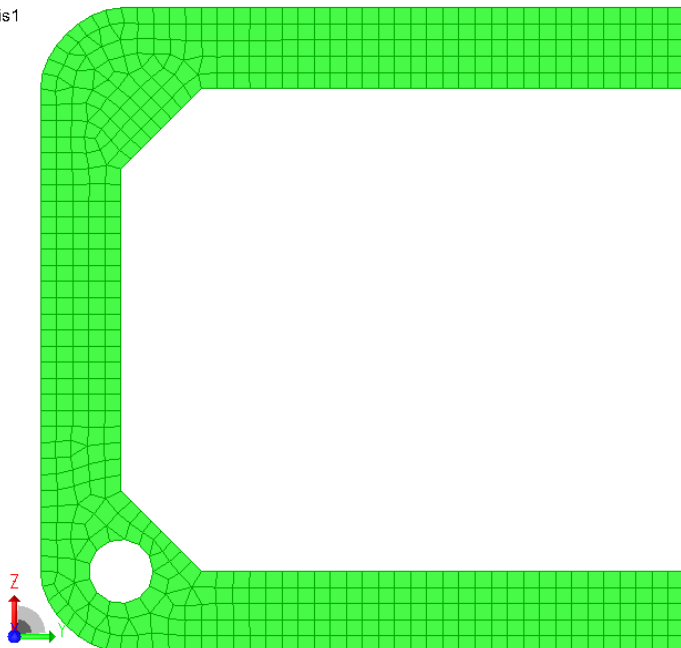
The mesh to the right has been generated using a local mesh density (0.25m) and the Advancing Front Mesher.

When you are satisfied with this mesh and you want to use it for all positions where this web-frame occur the mesh is locked by selecting the mesh, **RMB** and *Mesh Locking*.

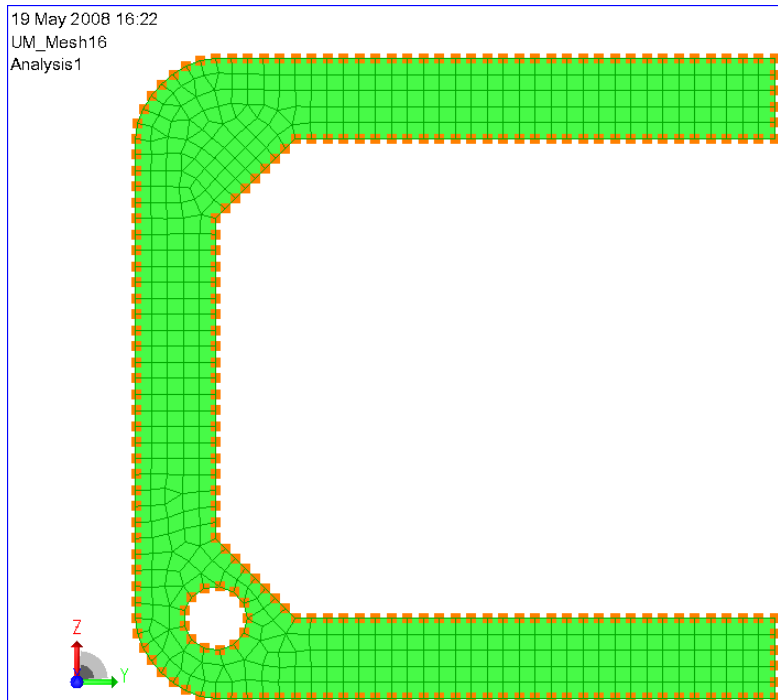
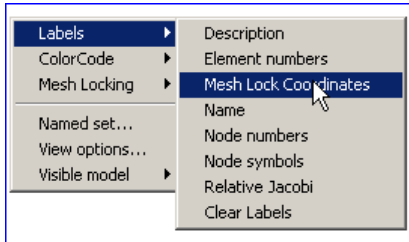


Mesh lock co-ordinates may be unlocked from same menu as above.

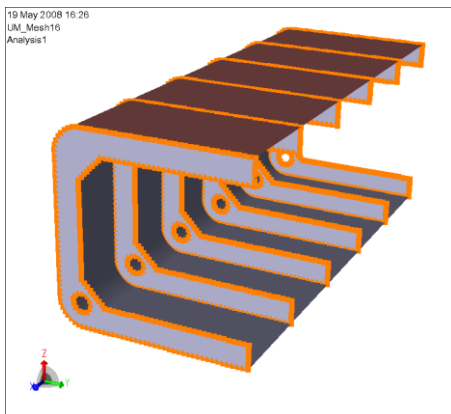
19 May 2008 16:17
UM_Mesh16
Analysis1



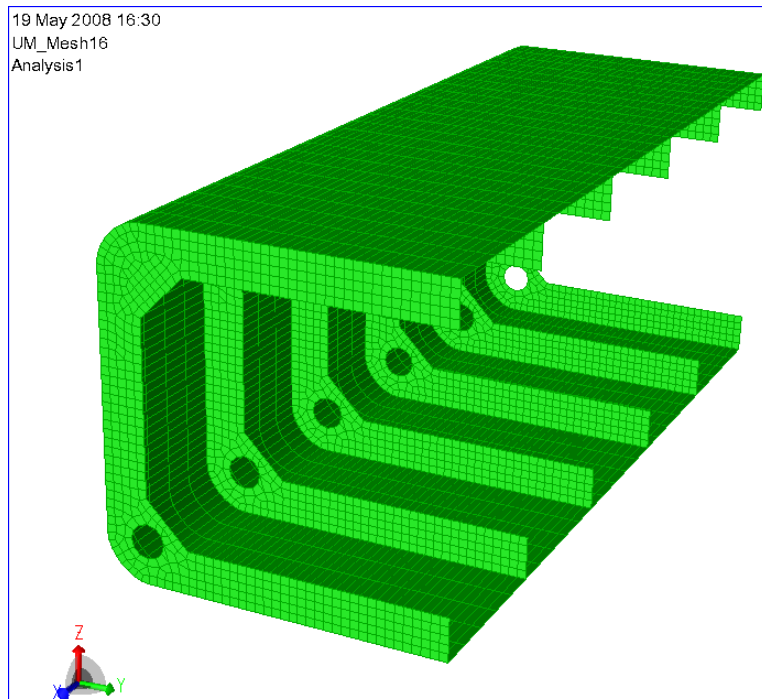
The co-ordinates of the mesh that has been locked may be labelled. Select the mesh, **RMB** and *Mesh Lock Coordinates*.



When the web-frame is copied to the five new positions, the mesh lock co-ordinates are part of the copy operation.



As can be seen, the mesh for each web-frame is identical.

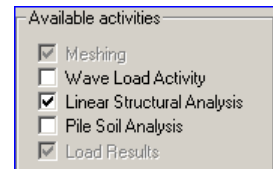


7. RUN ANALYSIS

The previous Chapters describe how to make a concept model consisting of structure, loads and boundary conditions as well as how to create analysis models (or finite element models). This Chapter addresses how to run analysis.

In GeniE it is possible to run the following analysis types using pre-defined analysis activities:

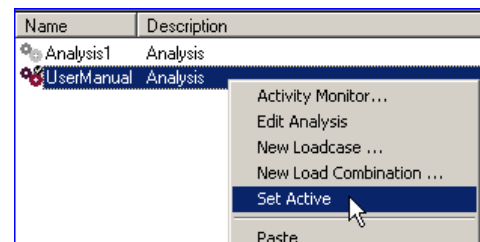
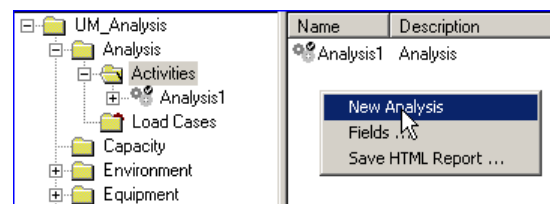
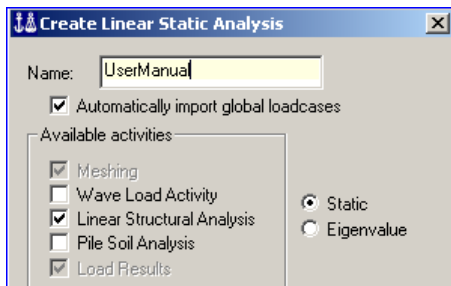
- Linear static
- Linear eigenvalue analysis
- Linear structural analysis including wave loads
- Linear structural analysis including wave loads and non-linear pile/soil analysis



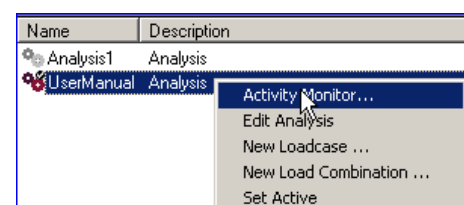
The above analysis requires that the model to be analysed is a so-called first level superelement. For all other analysis types, these can be done with ease in Brix Explorer by using pre-defined or your own workflows. A typical example may be a superelement analysis or workflows involving models created in GeniE used by Presel, HydroD, Sestra and Xtract.

7.1 Pre-defined analysis activities

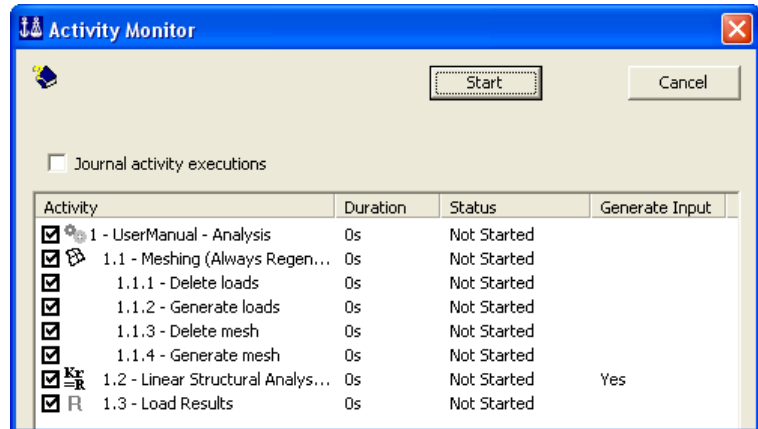
Analysis activities are defined from the browser as shown. You may have many analysis activities and you choose which one is active by setting it to “Active”. Please observe the tick-off option for “automatically import global loadcases”; if you want only the load cases defined in the activity folder this option must be ticked off. See also Chapter 4 for more information.



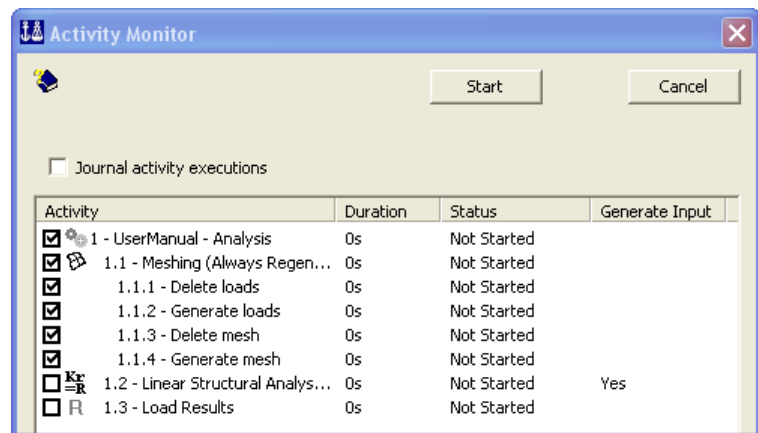
An analysis is started from **Tools/Analysis/Activity Monitor**, **ALT+D** or from the browser. For a linear structural analysis the activities included are meshing, linear structural analysis and load the results.



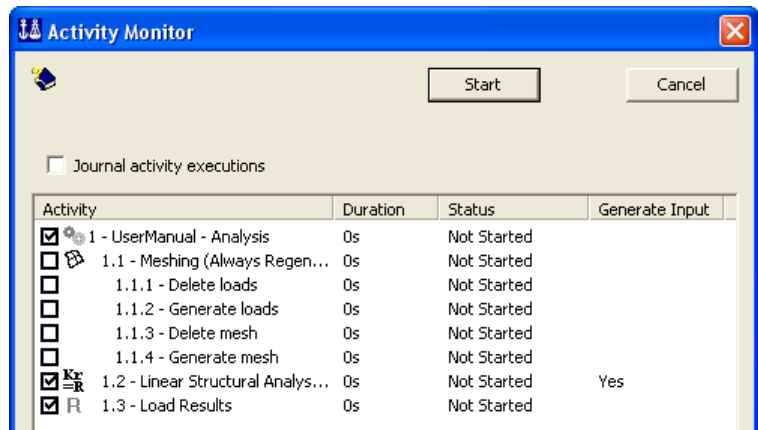
Please observe that it is possible to add the analysis activity to the journal file; in such case the analysis will be executed when you import the journal file.



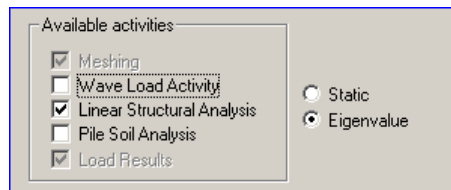
You can choose to run parts of the activities by de-selecting the actual activity. In the example to the right only a finite element model (a mesh) is created.



Similarly, if you want to use the existing finite element model it is not required to run the mesh activity over again. In this case you need to be aware that any changes to the concept model are not part of the analysis.



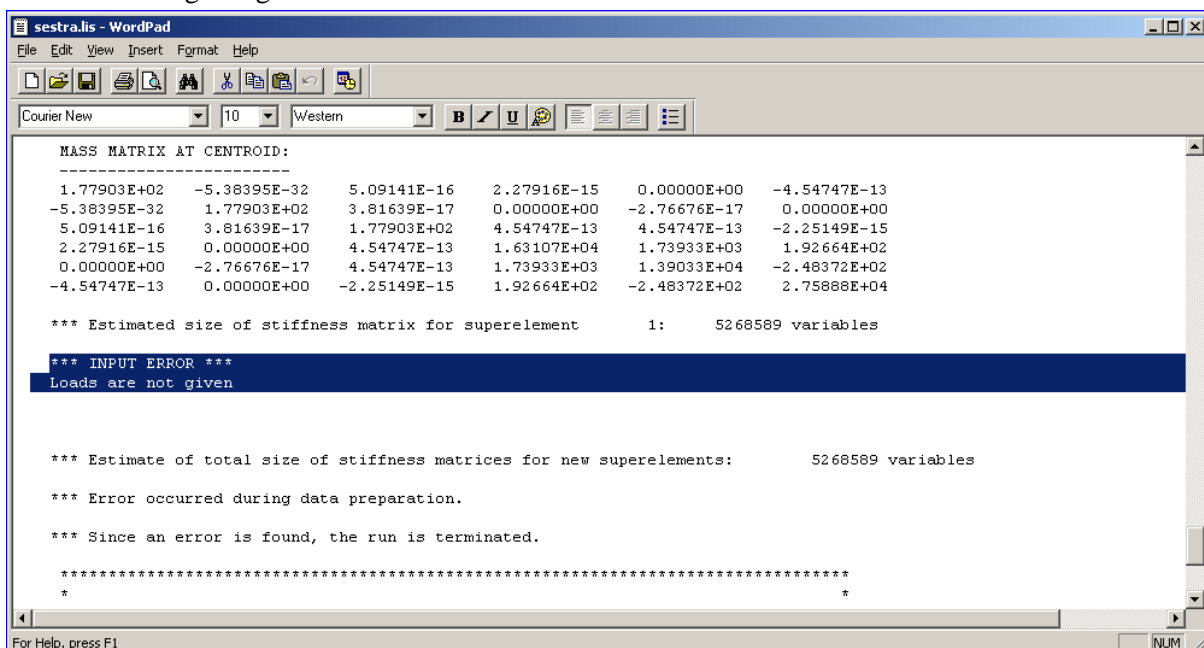
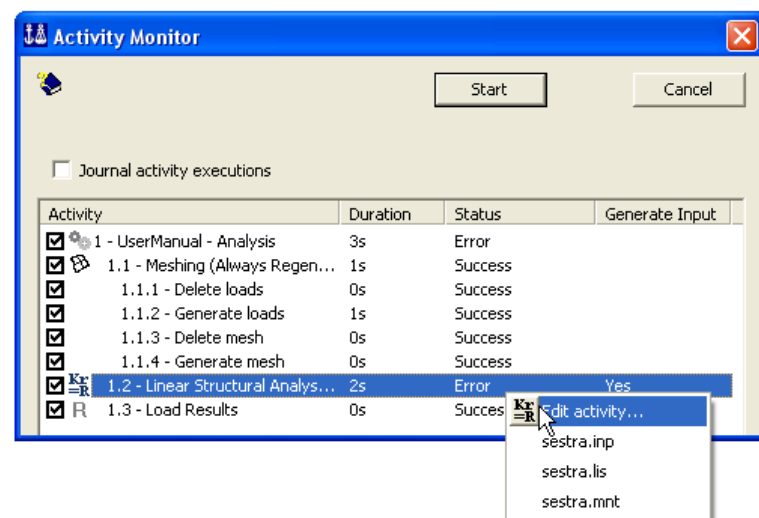
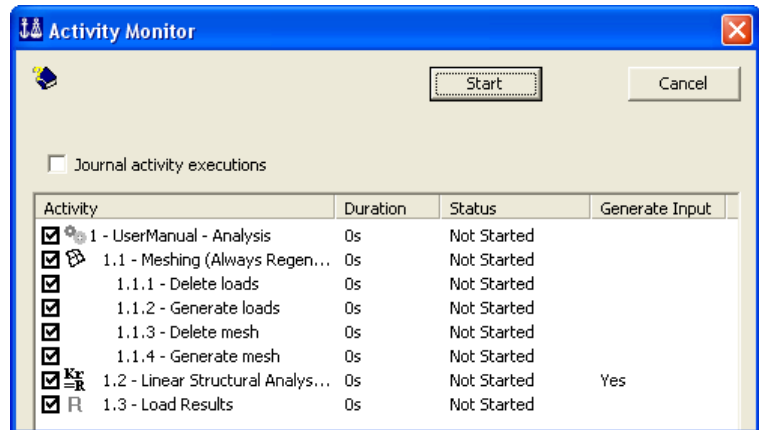
Analysis activities for eigenvalue analysis are created as follows:



The remaining analysis activities for wave and pile/soil analysis are not relevant for plate and shell structures. More information about these analysis activities can be found in the User Manual Vol I and II.

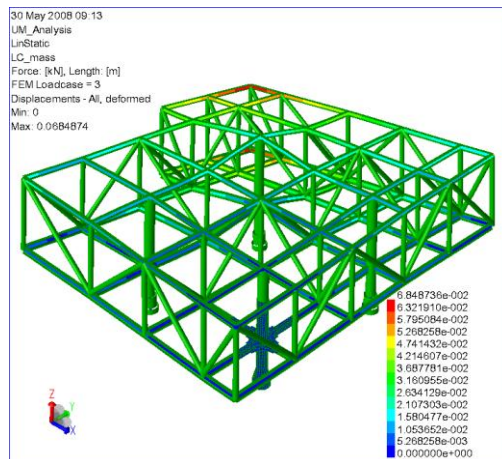
When an analysis is performed, the status field in the activity monitor contains relevant information. Typically; if the analysis is successful or if there is warning/error messages.

To find more information about the problem, **RMB** the relevant activity, select the listing file or maintenance file and search for the error message. In this case, the Sestra listing file is searched for the expression "error". Notice that the default editor is MS Notepad. In case you have associated another editor with the file type *.LIS this will overrule the default editor. The example below shows the Sestra listing using WordPad.

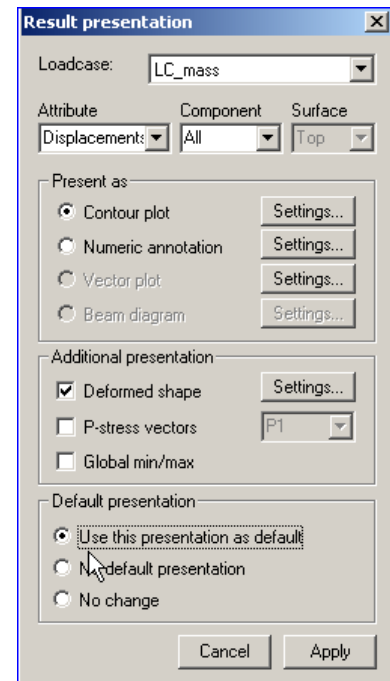


When the analysis is successful results can be viewed from the view “Results-All” or “Results – with Mesh”.

If you have not specified default result view you need to do this to see some results when setting the view to one of the result views. The default presentation view can be set from **Tools/Analysis/Presentation** or from **ALT+P**.



Activity	Duration	Status	Generate Input
<input checked="" type="checkbox"/> 1 - UserManual - Analysis	4s	Success	
<input checked="" type="checkbox"/> 1.1 - Meshing (Always Regen...	1s	Success	
<input checked="" type="checkbox"/> 1.1.1 - Delete loads	0s	Success	
<input checked="" type="checkbox"/> 1.1.2 - Generate loads	0s	Success	
<input checked="" type="checkbox"/> 1.1.3 - Delete mesh	1s	Success	
<input checked="" type="checkbox"/> 1.1.4 - Generate mesh	0s	Success	
<input checked="" type="checkbox"/> 1.2 - Linear Structural Analys...	3s	Success	Yes
<input checked="" type="checkbox"/> 1.3 - Load Results	0s	Success	

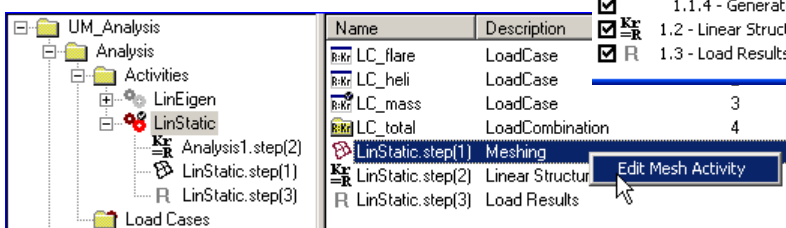


See the next Section for more information on how to present results.

7.2 Edit analysis activities

The pre-defined analysis activities can be edited from the activity monitor view or from the browser.

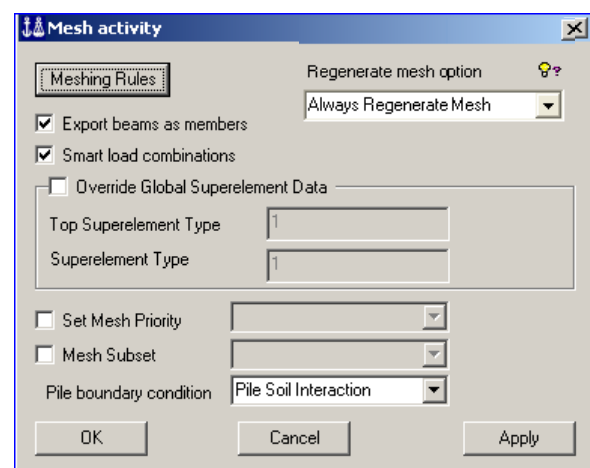
Both meshing options and specific run-time parameters can be modified. For meshing options, select the Activity “Meshing”, **RMB** and **Edit Activity**.



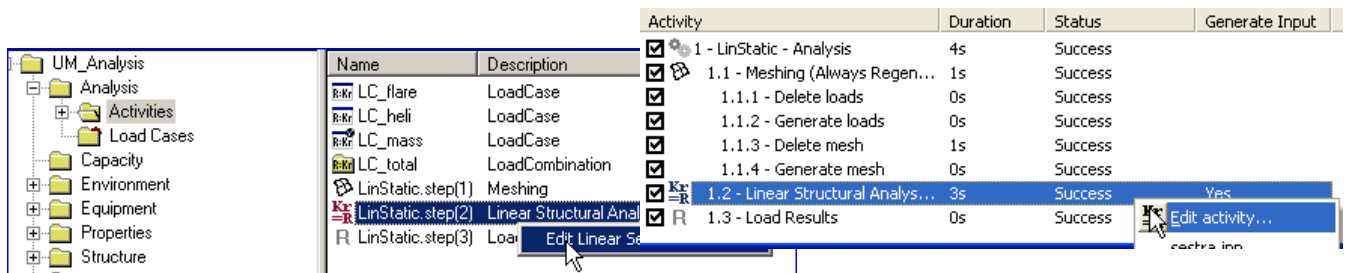
Activity	Duration	Status	Generate Input
<input checked="" type="checkbox"/> 1 - LinStatic - Analysis	4s	Success	
<input checked="" type="checkbox"/> 1.1 - Meshing (Always Regen...	1s	Success	
<input checked="" type="checkbox"/> 1.1.1 - Delete loads	0s	Success	
<input checked="" type="checkbox"/> 1.1.2 - Generate loads	0s	Success	
<input checked="" type="checkbox"/> 1.1.3 - Delete mesh	1s	Success	
<input checked="" type="checkbox"/> 1.1.4 - Generate mesh	0s	Success	
<input checked="" type="checkbox"/> 1.2 - Linear Structural Analys...	3s	Success	Yes
<input checked="" type="checkbox"/> 1.3 - Load Results	0s	Success	

The main parameters are:

- Export beams as members. If not selected the FEM file will not contain member information, but only pure finite element data
- Smart load combinations. In this case the load combinations are not transferred to analysis – the load combination is done in GeniE (or Platework, Framework or Xtract) after analysis.
- The “Set Mesh Priority” and “Mesh Subset” options are described in Chapters 6.5 and 6.6.

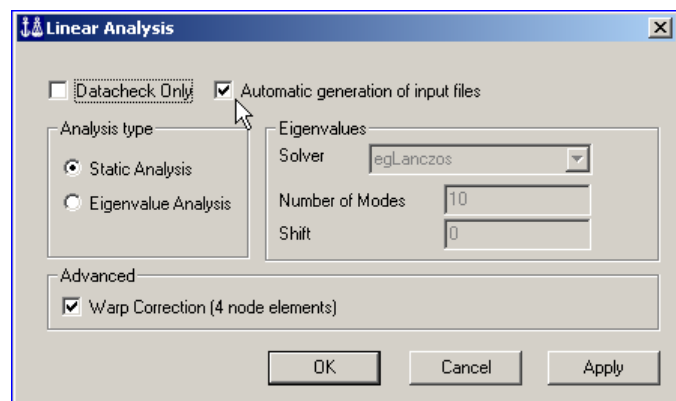


The run time parameters for a linear analysis can be modified from the activity monitor or the browser:



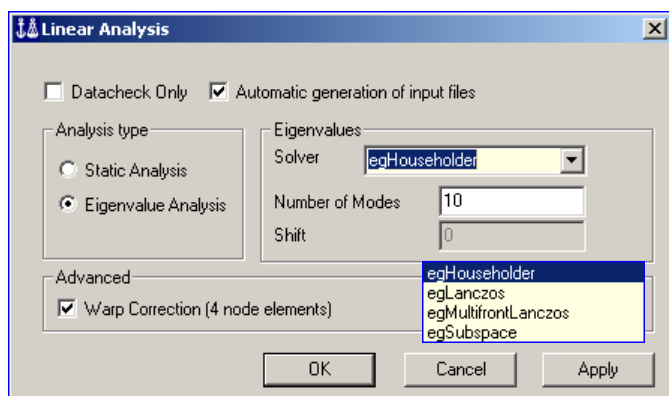
The parameters are:

- Data check only. This will check the data input and no analysis is carried out.
- Automatic generation of files. Input data to Sestra analysis is made by GeniE. In case you have deselected this option, you can make your own input file. See below how to do this.

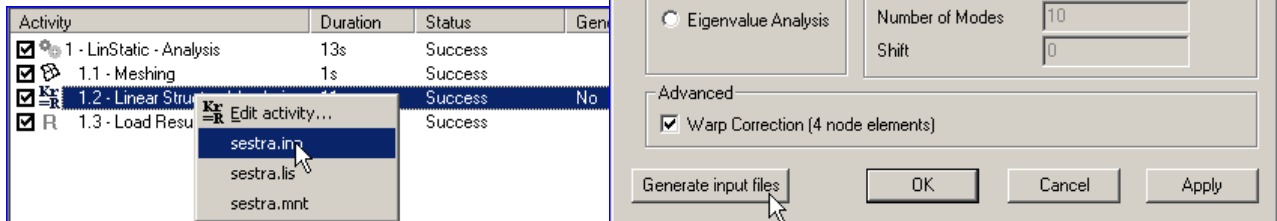


- Warp correction. The low order elements in SESTRA are flat. Flat quadrilateral shell elements may be impossible to construct over doubly curved surfaces. The condition of self equilibrium is not satisfied when the elements are not flat or warped. Self equilibrium in warped element may be achieved a posteriori for elements developed with respect to a flat geometry through the construction and application of an appropriate projection matrix

- Eigenvalue analysis. It is possible to define solver type and the number of modes. For the Multifront Lanczos solver the shift parameter can also be specified. For more details, please consult the Sestra User Manual



If you want to create your own input file to the Sestra analysis, deselect the “Automatic generation of input files”, click on “Generate input files” and edit the file from the activity monitor or browser.



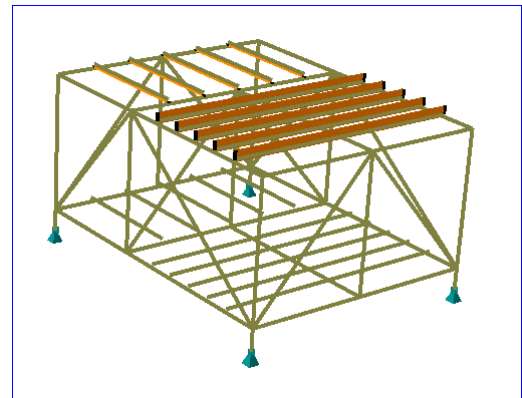
Any modifications you make to the Sestra input file will now be kept and used until you activate “Automatic generation of input files”. The input parameters to Sestra are described in the Sestra User Manual.

7.3 Static, eigenvalue and dynamic analysis

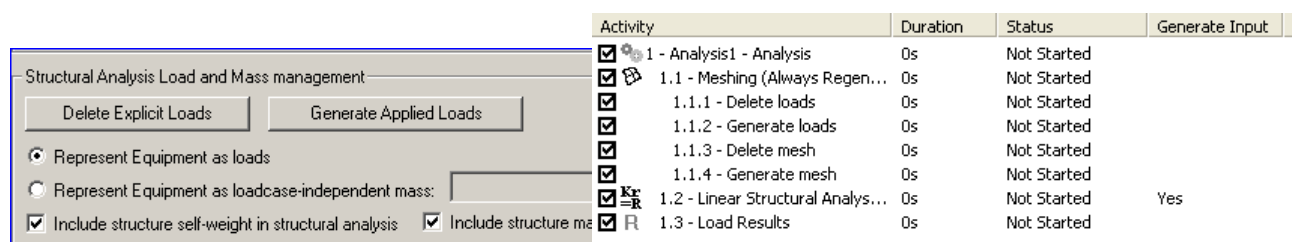
Sestra is capable of solving static, eigenvalue and dynamic analysis.

- A static analysis is based on structure, boundary conditions and loads. The loads may be of type manually applied loads, wave loads or inertia loads.
- For an eigenvalue analysis (also known as free vibration analysis), the input parameters are structure, boundary conditions and masses. The mass matrix is build up from the structural mass, the added mass from wave analysis and any explicitly masses (point masses or equipments).
- For dynamic analysis (or dynamic forced response analysis) there are two options; modal superposition in frequency and time domain, direct frequency response method or direct time integration.

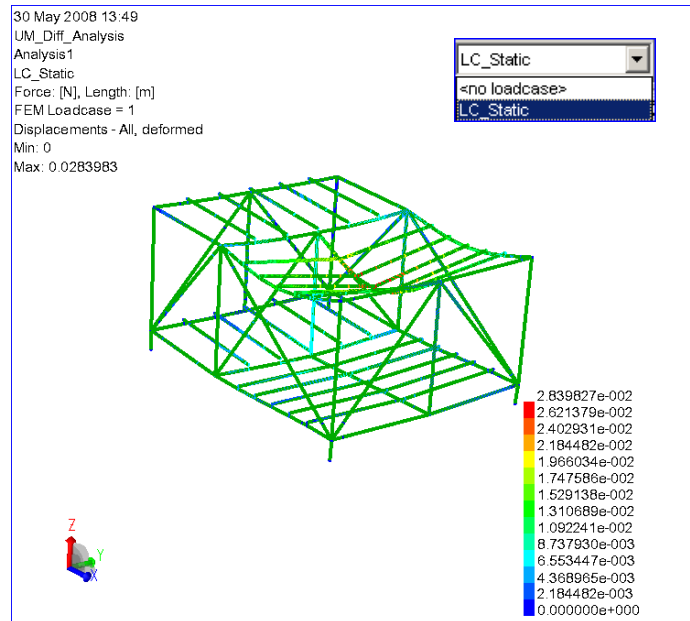
GeniE handles the two first alternatives directly from pre-defined analysis activities. It is also possible to do direct time integration by editing the input file to Sestra (notice that result presentation must be done in Xtract). To exemplify these options, a common model is used. The model has structural mass in addition to a load case with line loads.



The linear structural analysis is performed with the loadcase including the inertia load (gravity) and the line loads.



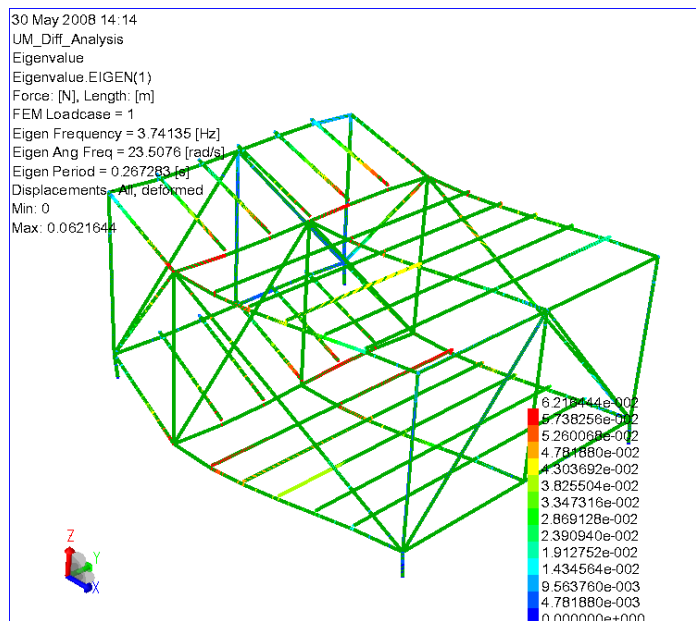
This will result in one result loadcase.



For an eigenvalue analysis, the loadcases will not create results. The eigenvalues are presented as resultcases. After the analysis you will get N number of result cases, numbered 1-N, where N is the number of eigenmodes computed. To avoid overlap between eigenvalue result-cases, the previous loadcase should either be deleted or be given a finite element loadcase higher than number of eigen-modes computed. Alternatively, make a new analysis activity and not import global loads.

By using the default input parameters the Lanczos solver is used to compute the ten first eigen-modes. The shape of the first eigen-mode is shown to the right. More information can be found in the analysis listing file.

Activity	Duration	Status	Generate Input
<input checked="" type="checkbox"/> 1 - Analysis1 - Analysis	0s	Not Started	
<input checked="" type="checkbox"/> 1.1 - Meshing (Always Regen...	0s	Not Started	
<input checked="" type="checkbox"/> 1.1.1 - Delete loads	0s	Not Started	
<input checked="" type="checkbox"/> 1.1.2 - Generate loads	0s	Not Started	
<input checked="" type="checkbox"/> 1.1.3 - Delete mesh	0s	Not Started	
<input checked="" type="checkbox"/> 1.1.4 - Generate mesh	0s	Not Started	
<input checked="" type="checkbox"/> 1.2 - Linear Structural Analys...	0s		<input checked="" type="checkbox"/> Edit activity...
<input checked="" type="checkbox"/> 1.3 - Load Results	0s		<input checked="" type="checkbox"/> sestrinp



NO.	EIGENVALUE UNIT: (SEC) ⁻²	FREQUENCY UNIT: HERTZ	PERIOD UNIT: SEC
1	0.5526082E+03	3.741	0.26728
2	0.1176054E+04	5.458	0.18322
3	0.2284700E+04	7.607	0.13145
4	0.4049356E+04	10.128	0.09874
5	0.4390217E+04	10.545	0.09483
6	0.5155087E+04	11.427	0.08751
7	0.5433083E+04	11.731	0.08524
8	0.5571336E+04	11.880	0.08418
9	0.6450378E+04	12.782	0.07823
10	0.8076270E+04	14.303	0.06992

To do dynamic analysis using the direct time integration method it is necessary to edit the analysis input file. Remember to deselect the automatic generation of input file and manually create the input file.

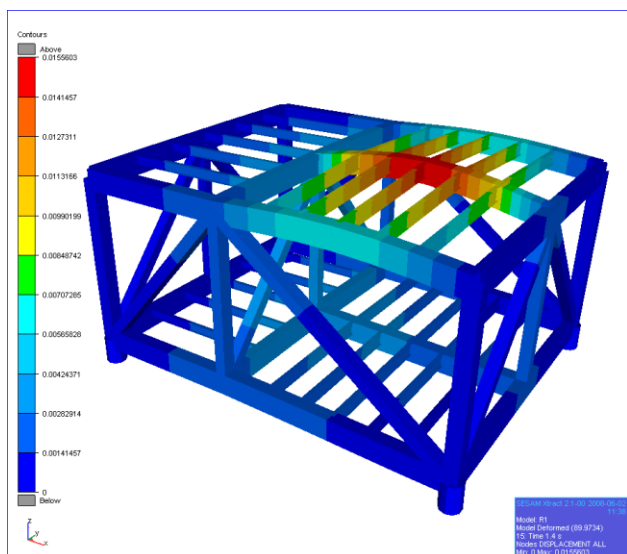
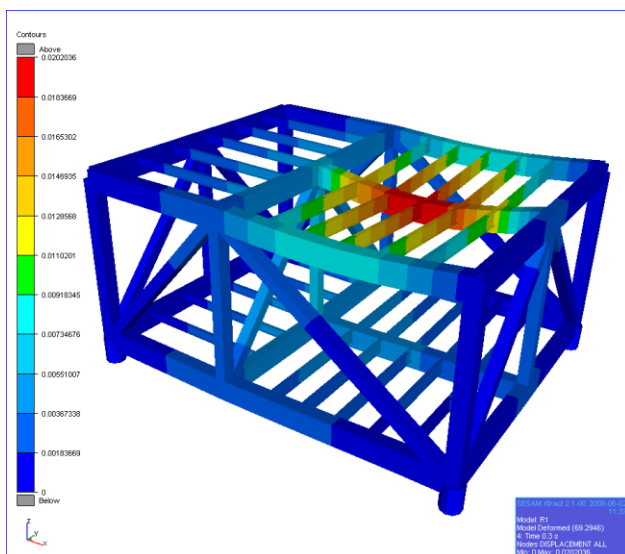
The load history used in this example is a sinusoidal load (360 degrees) over 2.4 sec. The corresponding input for the one load case (line loads) and the mass matrix is shown below.

```

sestra.inp - Notepad
File Edit Format View Help
HEAD
COMM
COMM      Created by: GeniE   D4.0-12   26-May-2008
COMM
COMM      Date : 01-Jun-2008   Time : 10:32:35   User : nek
COMM      Manually edited
COMM
COMM-----
COMM CHECK ANTP      MOLO STIF RTOP LBCK      PILE      CSING      SIGM
CMAS   0.   2.      0.   0.   0.   0.      0.   0.      0.
RNAME 20080602_112757_ NORSAM
ITOP   1.
FRSP   4.
COMM LOAD HISTORY 1
LOHI   1.      1.   1.   1.   1.
LCOM   1.      1.      1.
COMM <-1-><--2--><--3--><--4--><--5--><--6--><--7--><--8-->
HIST   1.      1.      1.      0.000   0.259   0.500   0.707   0.866
      0.966   1.000   0.966   0.866   0.707
      0.500   0.259   0.000   -0.259   -0.500
      -0.707   -0.866   -0.966   -1.000   -0.966
      -0.866   -0.707   -0.500   -0.259   0.000
TILO   1.      1.      0.0   0.1   0.2   0.3   0.4
      0.5   0.6   0.7   0.8   0.9
      1.0   1.1   1.2   1.3   1.4
      1.5   1.6   1.7   1.8   1.9
      2.0   2.1   2.2   2.3   2.4
TOUT   1.      2.      0.      0.1   2.3
COMM RTRAC PRNT STOR EQUI SEL1 SEL2 SEL3 ...
RETR   3.      0.   0.   0.   0.   0.   0.
IDTY   1.
DYMA   1.
Z

```

The results can be seen in Xtract. The pictures below show deformations at time steps 0.3 sec and 1.4 sec.



8. RESULTS PROCESSING

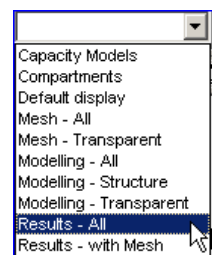
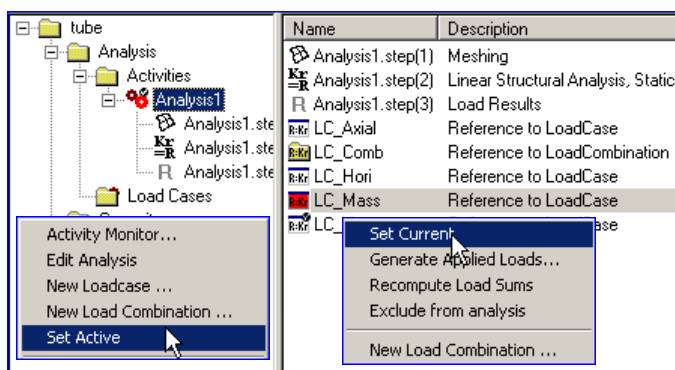
GeniE can do

- General post-processing – deformations, plate and beam stresses. For more detailed processing like scanning Xtract can be used.
- Code checking of beams according to offshore standards
- Code checking of plates according to the CSR Bulk standard

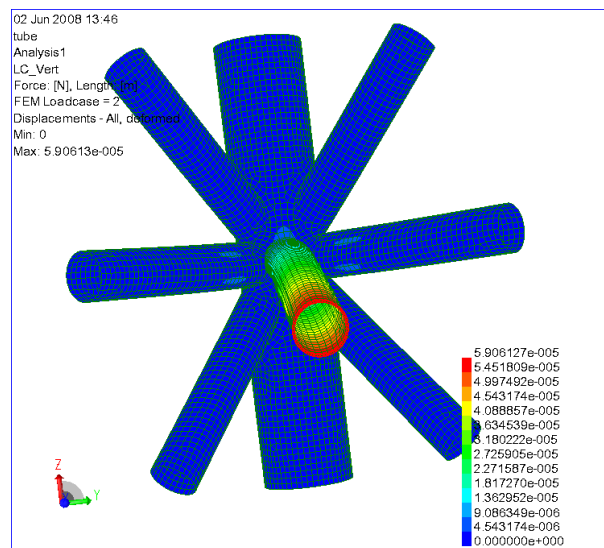
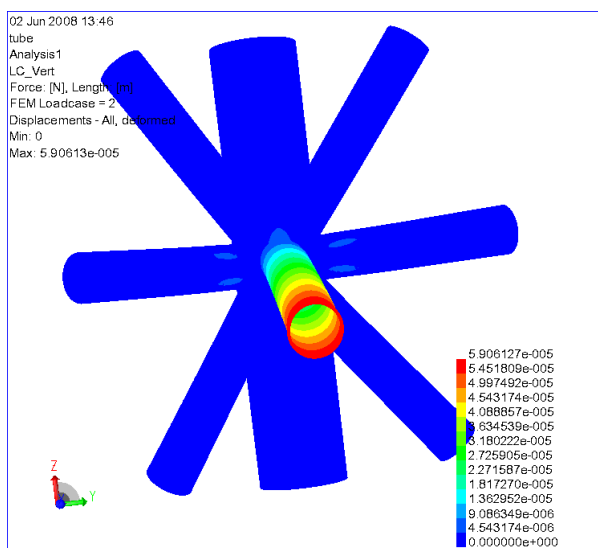
This Chapter gives a short introduction on how to do general post-processing, for more details Volume I of the User Manual should be consulted. Code checking of beams and plates is documented in Volume IV and V of the User Manual.

To view results you can use one of the pre-defined views “Results – All” or “Results – With Mesh”.

Prior to any result viewing you need to set a load case to “Current”. In case you have several analysis activities you also need to ensure that the right one is set to “Active”.



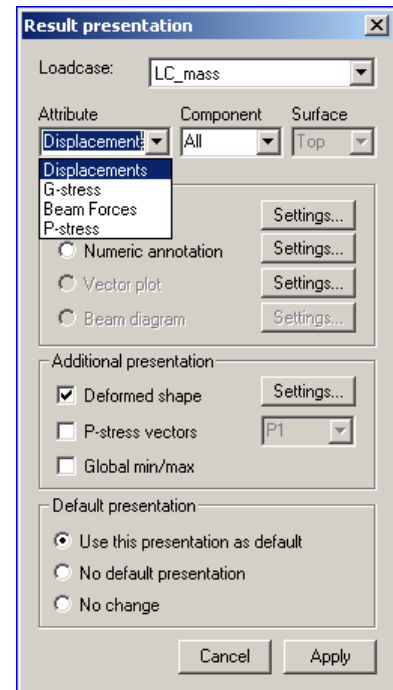
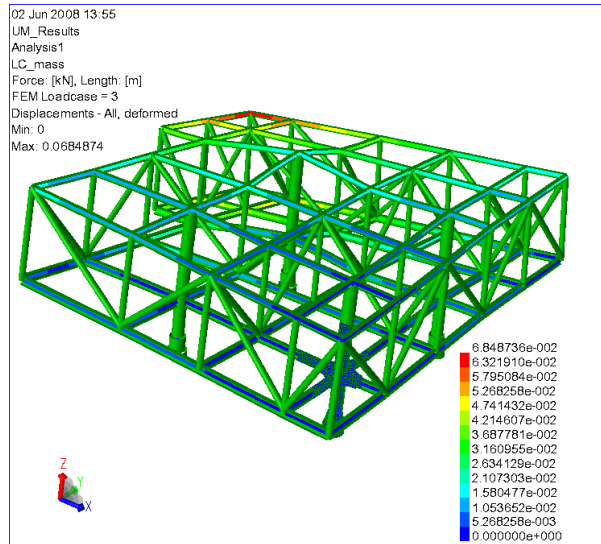
The view “Results – All” is a pure results view, while the other view includes the finite element mesh.



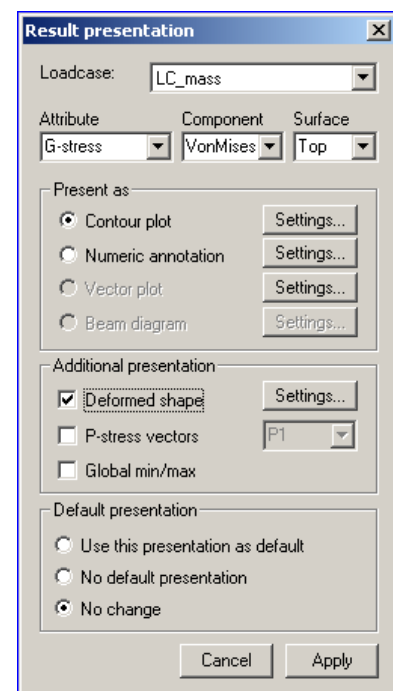
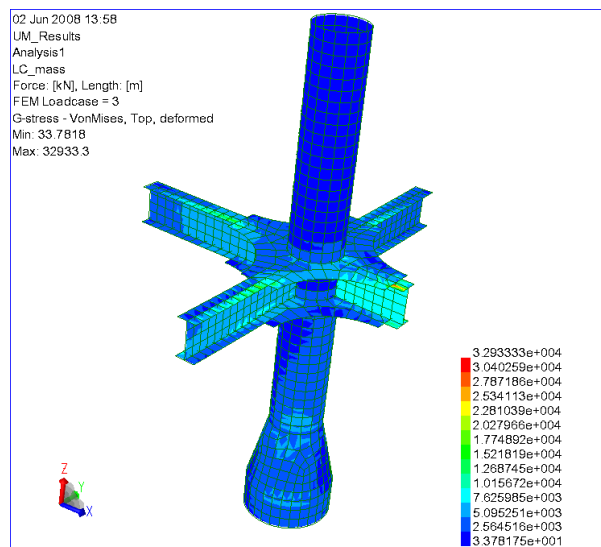
The result attribute can be changed from *Tools/Analysis/Presentation* or **ALT+P**.

You should specify which attribute you want to use as your default view. The default value is project specific, i.e. you need to set the default for each workspace.

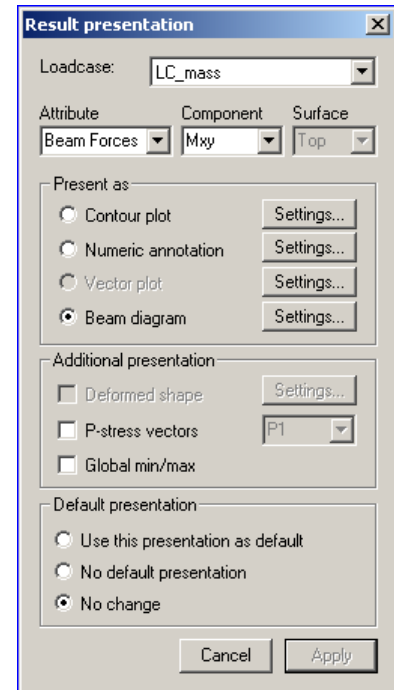
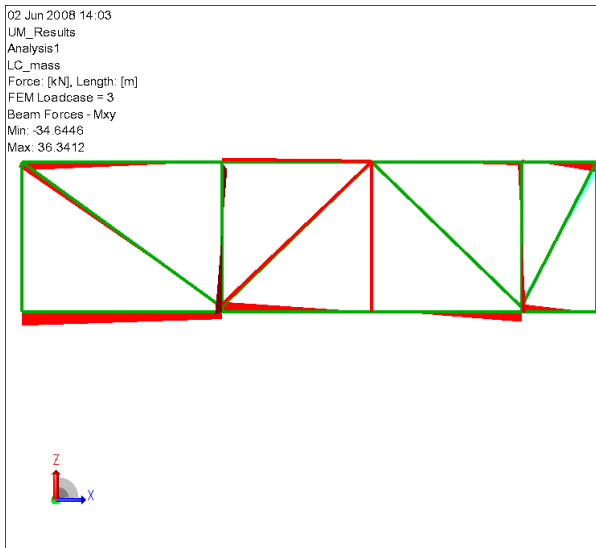
The settings to the right generate the results as shown below.



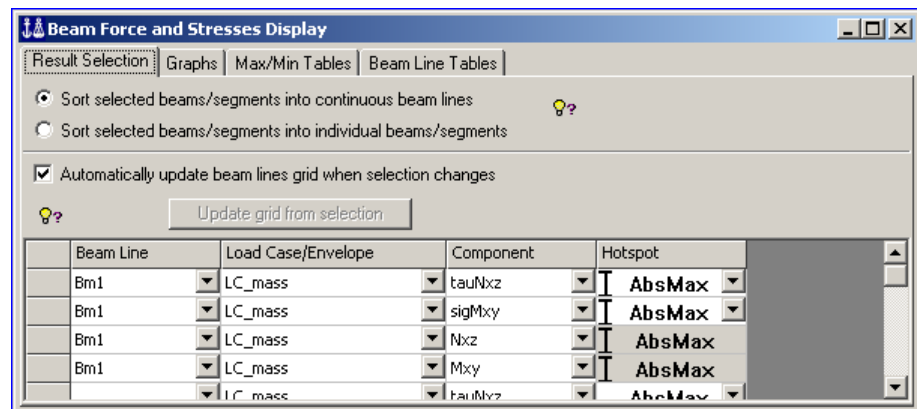
By selecting parts of the structure and modify the attribute to stresses, another result view is generated (if needed, remember to click the exclamation mark to update the result).



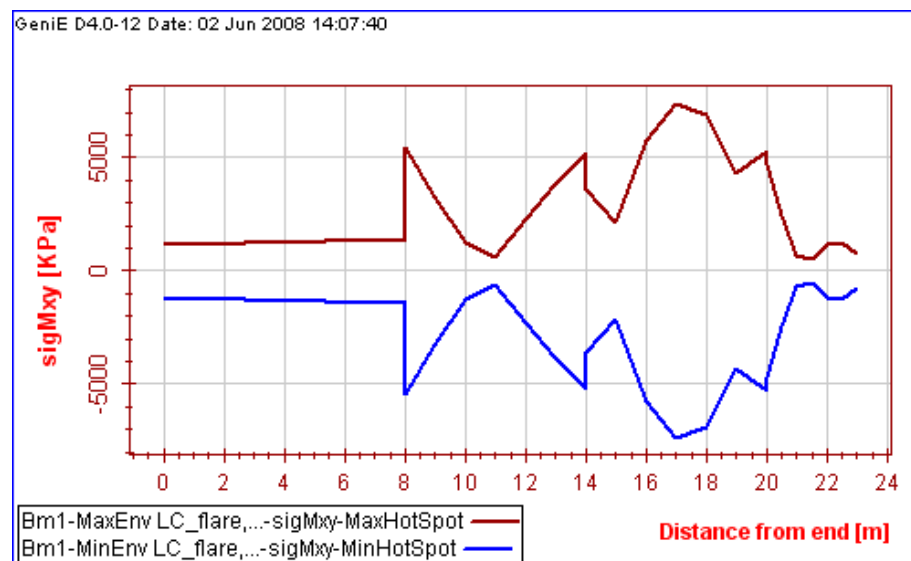
Beam forces can be viewed in a 3D and 2D view. The 2D view is also used to look at beam stresses. In the first case the entire structure or parts of it is used. In this example one relevant row is selected and beam moments are shown.



To use the 2D view, select a beam or beams and activate the functionality from **Tools/Analysis/Beam Force/Stress Diagram**. You are now prompted for type of attribute and loadcase to use – for more details, please consult the User Manual Vol IV.



The example to the right shows a typical envelope for bending moments.



9. IMPORT AND EXPORT

Import of data can be done for various data types. These may be defined by previous GeniE sessions or by other systems. There are two types of import; the journal file that can be imported from **File/Read Command File** or other formats that can be imported from **File/Import**.

The journal file is automatically created during a modelling session. It can also be edited and there is an option to make a condensed journal file without any history (also known as a “clean” journal file) from **File/Export/GeniE journal file**.

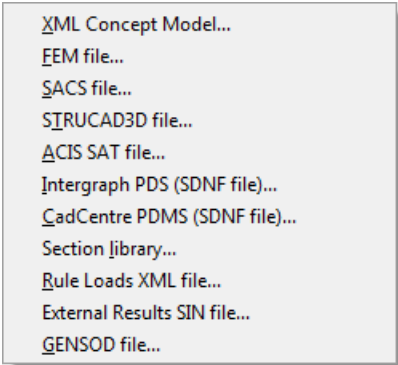
The data sources that can be imported are:

- XML describing the concept model. This file is normally generated by GeniE using an export option.
- FEM file (Sesam’s neutral file format for finite element models)
- SACS input file
- STRUCAD3D input file
- ACIS SAT format (shells only)
- Intergraph PDS (sdnf)
- Cadcentre PDMS (sdnf)
- Section libraries (see also Section 3.1.2)
- Rule loads XML file. This file is defined by Nauticus Hull in accordance with the CSR rules for bulk ships. The file contains loads and boundary conditions.
- External Results SIN file (Sesam’s neutral file format for finite element results). Result files from other SESAM programs can be imported for post processing in GeniE. See User Manual Volume IV for details.
- The GENSOD file, GENSOD.inp, stores the soil data needed for by SPLICE

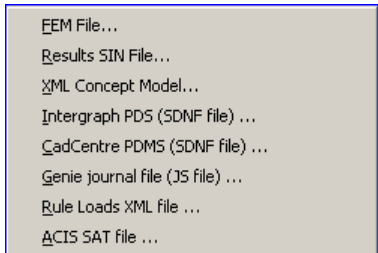
Similarly, export of data can be done from File|Export. The following export operations can be done.

- FEM file. Export and rename a FEM file
- Results SIN File. Export and rename a SIN file
- XML Concept Model. Create and export a neutral model file. See below for more information.
- Intergraph PDS (sdnf)
- CadCentre PDMS (sdnf)
- GeniE Journal file (JS file). Create a “clean journal file” without historical data. See below for more information.
- Rule Loads XML file. Export necessary data to Nauticus Hull to generate loads and boundary
- ACIS SAT file. Export surface data that can be accessed by other programs supporting the format.

The purpose of export & import of the command file, the xml concept model file and the FEM file is described in the following; for more details on the other formats, please consult User Manual Vol. I.



XML Concept Model...
 FEM file...
 SACS file...
 STRUCAD3D file...
 ACIS SAT file...
 Intergraph PDS (SDNF file)...
 CadCentre PDMS (SDNF file)...
 Section library...
 Rule Loads XML file...
 External Results SIN file...
 GENSOD file...

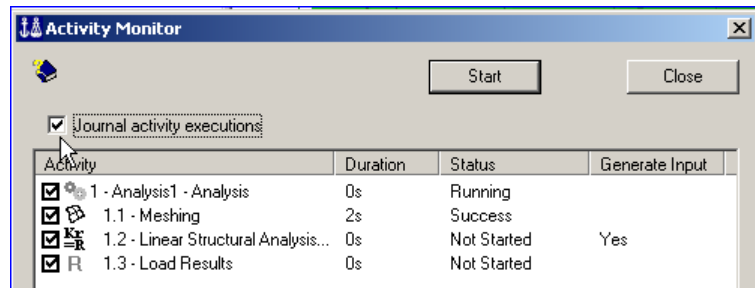


FEM File...
 Results SIN File...
 XML Concept Model...
 Intergraph PDS (SDNF file) ...
 CadCentre PDMS (SDNF file) ...
 GeniE journal file (JS file) ...
 Rule Loads XML file ...
 ACIS SAT file ...

This Chapter focuses primarily data storage of data produced by GeniE. It is important to know the limitations in the journalling system and the xml neutral file to ensure re-use of data for new projects and when upgrading to newer versions of GeniE.

9.1 The command file generated by GeniE

All operations except for graphical manipulations and picture generation are automatically logged. For some operations like running analysis, export finite element data and report generation you need to specify that you want to log the operation. An example of such is shown to the right.



When importing the command file to a new workspace the same model will be recreated. If you want to base your data exchange using this option you should check that the js-file is running safely at all times. To do this, you should import the journal file several times to a dummy workspace during the modelling of your real project to ensure that it will give the desired result. You can also add comments to the journal file by editing the file or by adding these in the command line interface window.

It is also possible to include operations that are not logged. A typical example can be to specify a view angle and to make a picture of it. For more details, see Chapter 5 of the User Manual Volume I.

9.2 The condensed command file

The purpose of the “clean” journal file is to support import and export of:

- For frame models with simple plates (planar, straight edges, no holes)
- Wave load and soil pile analysis runs
- Line and point loads, simple surface loads
- Equipment
- Environment
- Guiding geometry
- Sets
- For regression runs, where the input file must be editable

The following limitations apply:

- Curved geometry and advanced surface loads are not exported.
- Capacity checks are not exported.
- Reports are not exported.

9.3 The XML concept model file

The purpose of the xml concept model file is to support import and export of beam and/or shell/plate structures.

- For migration of model between GeniE versions
- For repeated analyses, where editing the input file is not required

The following is supported:

- The activities under an analysis, i.e. the local settings on the meshing activity, wave load run activity and analysis run activity
- Load combinations including wave load cases from an analysis.
- Loads of all kinds
- Equipment
- Environment
- Compartments
- Guiding geometry
- Sets
- Settings from *Edit|Rules|...* are exported

The following limitations apply:

- Capacity checks are not exported
- Reports are not exported
- Dynamic sets are exported as normal sets
- Dummy hydro pressure loadcases are not exported

Before exporting an xml file you should verify your model by clicking the “Verify Model” button. This will reveal whether the model contains any geometry or topology errors (ACIS errors).

If your model contains ACIS errors it might give you problems if you try importing the xml back into Genie later.

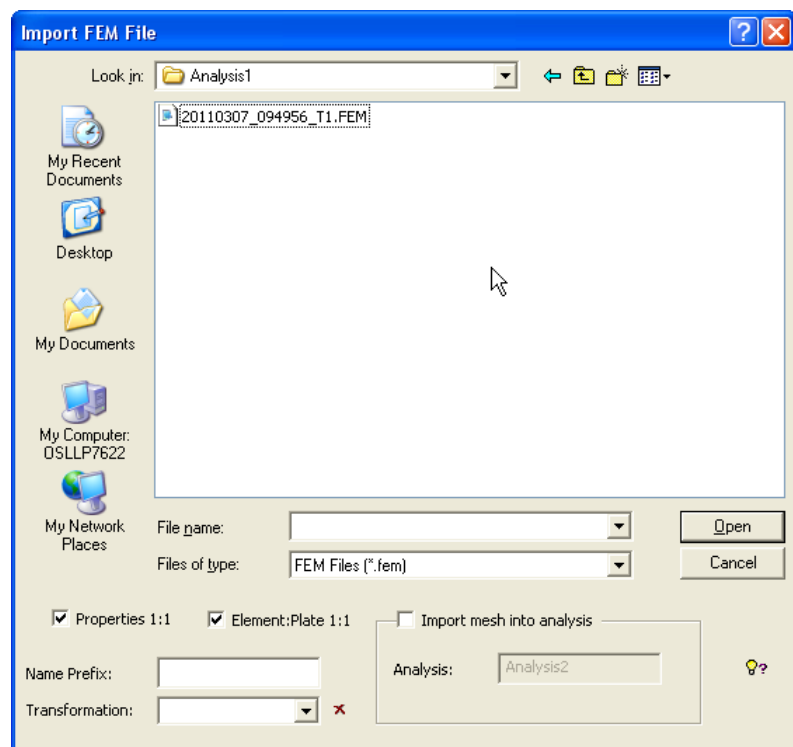
9.4 From/to FEM

When importing a FEM file (created by e.g. Preframe, Prefem or Patran-Pre) the finite element model is converted to a concept model. For beams there is a 1:1 relation between a beam finite element and a beam concept. There is a name recognition whereby a beam finite element number 25 becomes e.g. Bm25. If you want higher level concepts (for example a continuous leg) then select the beam concepts, RMB and *Join Beams*.

For plates GeniE will make the largest plate concept possible given that the plate finite elements are in the same plane and have the same properties (e.g. thickness and material). If you want a 1:1 relation between plate finite element and plate concepts you must tick off this option in the import dialogue box.

When “Properties 1:1” is checked, GeniE imports all sections and materials according to the following priority: Any explicit name will be used (for example the FEM-file was created in another GeniE session, i.e. the FEM file contains concept information)

- Type prefix + section/material number (for example section number 23 on the FEM file becomes Sct23, in this case the FEM file has no concept information)
- Default name schema as found under **Tools/Customize/Default Names**



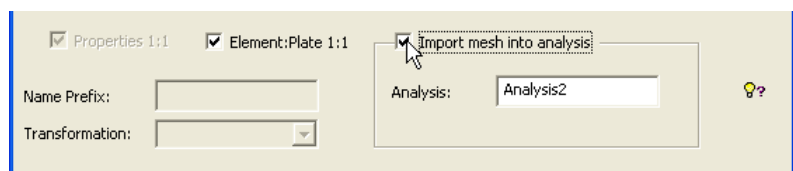
When “Import mesh into analysis” is checked, GeniE imports the FEM file and create an analysis activity for the imported model. This analysis will contain a meshing activity containing the

imported mesh. GeniE will also store a two way connection between the information on the FEM file and the concepts/properties they represent, so you can actually modify parts of your model (properties and loads) without having to completely remesh the model. Instead GeniE updates the properties for the modified elements and regenerates the loads if needed.

This implies that the imported external element/node numbers are kept and saves much time when working with large models. (When deleting point masses and boundary conditions, the internal element numbers may be rearranged.)

Properties that can be changed without changing the element/node numbers are:

- cross sections for beam



- thickness for plate
- materials
- explicit loads (only point and line loads on beams).
- Local coordinate system for beam
- Eccentricities.

Loads that can be changed without changing the element/node numbers are:

- Gravity loads
- Line loads on beams
- Point loads
- Node accelerations

Structure concepts that can be changed without changing the element/node numbers are:

- Point masses
- Support points

Some properties and loads cannot be imported into GeniE but will stay untouched on the FEM file as long as the mesh is not regenerated. If loads are added or modified all non-imported loads will be destroyed.

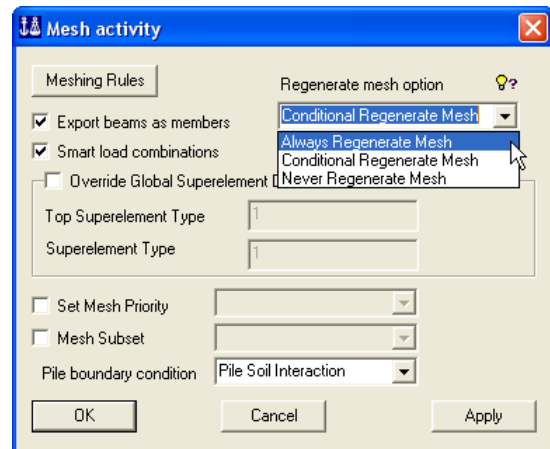
The following properties and loads are not imported into GeniE:

- Shim connections
- Linear dependencies
- Non structural plates
- Line loads on shells
- Surface loads on shells
- Temperature loads

Note that no higher order elements are supported with this import method.

Note that other changes will make it necessary to regenerate the mesh causing the element/node numbers to change. An example of such a change is adding a beam or a plate.

By default when importing a FEM file, the “Regenerate mesh option” is set to “Never Regenerate Mesh”. Under such circumstances, if you have done changes that make regenerating mesh necessary and then run the mesh activity by clicking on the button “Start” on the activity monitor, the generated FEM file will not reflect the changes that are made. However user can bypass this default setting by either change the “Regenerate mesh option” or explicitly type in the JS command in the GeniE command window.



The FEM import retrieves all data found on the FEM file, piles and overlapping beams included. For overlapping beams, the inner and outer beams should have the same FE mesh density prior to import.

Observe that when importing hinges these are subject to name mangling, i.e. equal hinges will be reused. Hinges modelled in e.g. Preframe are finite elements and when translating these to concepts in GeniE hinge properties will be created and connected to relevant beam ends. When e.g. 14 hinge elements are equal on the FEM file, they will appear as one single hinge property (name mangling).

The main purpose of the FEM file import is to import data from other SESAM programs and systems (where data is converted to FEM format prior to import in GeniE) and not to act as a neutral storage format. The reason for this is that the FEM file format does not include all concept information. For the purpose of neutral storage the journal file should be used. You may choose to use the journal file generated during modelling activities (including the history) or a clean variant of the journal file (without history), see Chapter 3.7.6.

To export a FEM file use command **Tools/Analysis/Export FEM**. This can be done after creating the mesh using the command **Tools/Analysis/Create Mesh**.

9.5 Import SACS file

SACS files can be imported to GeniE.

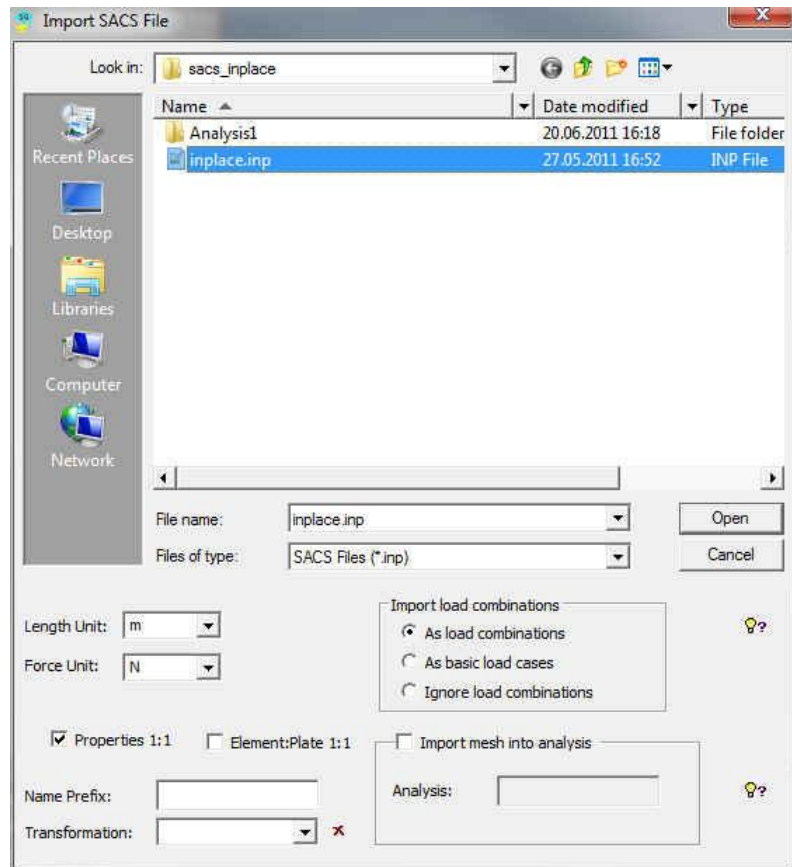
Properties 1:1

When “ Properties 1:1” is checked, GeniE imports all sections and materials according to the following priority: Any explicit name will be used.

- Type prefix + section/material number (for example section number 23 on the SACS file becomes Sct23, in this case the SACS file has no concept information)
- Default name schema as found under *Tools/Customize/Default Names*

Element:Plate 1:1

For plates GeniE will make the largest plate concept possible given that the plate finite elements are in the same plane and have the same properties (e.g. thickness and material). If you want a 1:1 relation between SACS element and plate concepts you must tick off this option in the import dialogue box.



Import load combinations

- As load combinations – import load combinations from a js file (sacstmp_CMB.JS) generated from the SACS inp file.
- As basic load cases – import load combinations as basic load cases. Note that currently recursive load combinations are not handled.
- Ignore load combinations – do not import load combinations.

Note that the file sacstmp_CMB.JS is generated independent of option selected.

Import mesh into analysis

- This option stores the imported SACS file as a FE model including its FE Node and Element numbering.
- See also chapter 9.4.

An overview of the SACS import can be found in the reference documents. There are some additions to the reference document valid for GeniE V6.0:

1. LDCOMB command

In case of GeniE load combinations are no longer written to '.CMB' file. Instead they are written to 'sacstmp_CMB.JS'. If the 'As load combinations' option is selected this file is automatically read as part of the import process. If the 'As basic load cases' option is selected the combinations are additionally converted to and imported from the temporary '.FEM' file. Note that recursive load combinations are currently not handled.

2. LCOMB command

This command comply to the format described in the SACS IV manual – a more recent format. The above description of the LDCOMB command is also valid here.

3. Extended built-in cross section library

The built-in library is extended with the following sections:

WF profiles (Euro) :				
IPE140				
IPEO240				
WF profiles (AISC) :				
W38X282				
WF profiles (other) :				
100X100	250X250	336X249	406X403	606X201
125X125	250X255	340X250	414X405	612X202
150X75	298X149	344X348	428X407	582X300
148X100	300X150	344X354	458X417	588X300
150X150	294X200	350X350	498X432	594X302
198X99	298X201	350X357	446X199	692X300
200X100	294X302	396X199	450X200	700X300
194X150	298X299	400X200	434X299	708X302
200X200	300X300	404X201	440X300	792X300
200X204	300X305	386X299	496X199	800X300
208X202	304X301	390X300	500X200	808X302
248X124	310X305	388X402	506X201	890X299
250X125	310X310	394X398	482X300	900X300
244X175	346X174	394X405	488X300	912X302
244X252	350X175	400X400	596X199	918X303
248X249	354X176	400X408	600X200	

4. Improved support of cross sections

PRI – Enhanced conversion, defining a Bar section (GBARM card) including stiffness properties. Previously a General section was defined.

ANG – Enhanced conversion, correct calculation of product of inertia about Y and Z axes

TEE – Support of T section. It is converted to an I section with bottom flange width equal to web thickness.

5. Model size limitations

For GeniE V6.00 - 64 bit version the max number of nodes and elements is 25000, while the limit is 10500 for the 32 bit version. For both 32 and 64 bit versions the number of load combinations that can be handled is 1000 and number of basic loadcases per load combination is 100.

9.6 The ACIS SAT file

The SAT file contains all the geometry in a model, all the plates and all the edges.

When Genie imports a SAT file it tries to “glue” all parts in the SAT file together to one valid model. If the points and edges in the SAT file are not placed with the required accuracy the import will fail. Two points in the SAT file can not deviate with more than 1e-6 m (The SAT file *must* always use m as unit) if they are to be treated as the same point.

The ACIS SAT file is normally compatible with previous versions. ACIS R19 Sp2 is used by Genie V5.

Genie does support the ACIS SAT format for geometry exchange, not IGES and others. For conversion you may use 3rd party packages.

The following is supported:

- Geometry is supported, but nothing else.

The following limitations apply:

Problems when importing from other systems with incompatible geometry, such as:

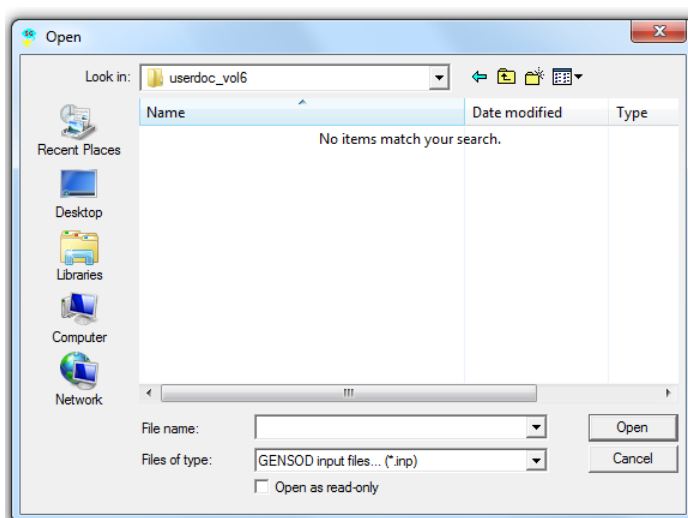
- Solid models

9.7 The GENSOD file

The stand-alone application GENSOD generates the soil data needed by SPLICE, storing them in GENSOD.inp.

A GENSOD.inp file can be imported into GeniE.

GeniE restores the information from the GENSOD.inp file by reproducing the js-commands, thus supplying the folders “Environment/Location” and “Environment/Soil” of the GeniE browser with the relevant data.

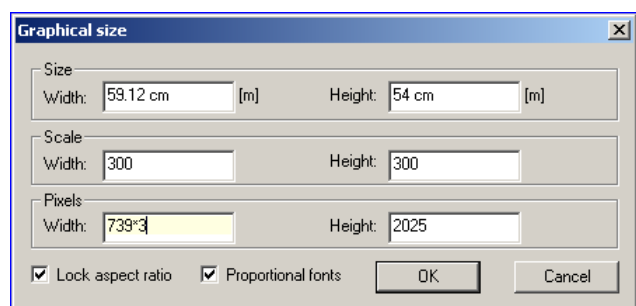
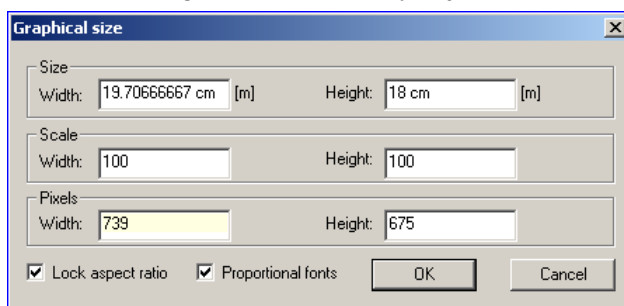


10. PICTURES AND REPORTS

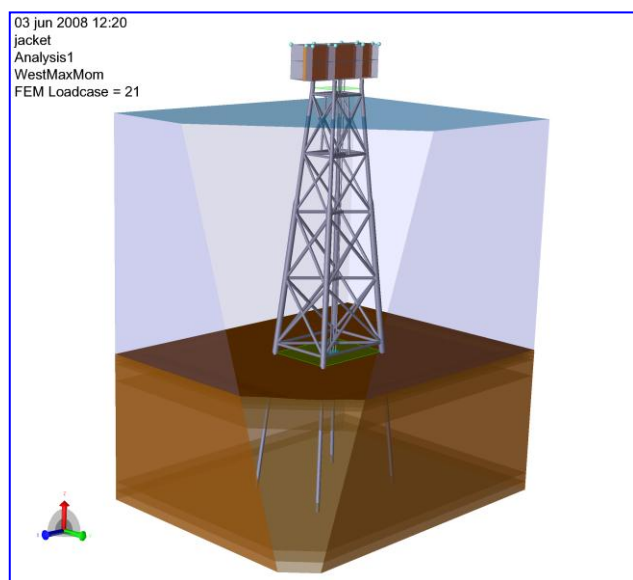
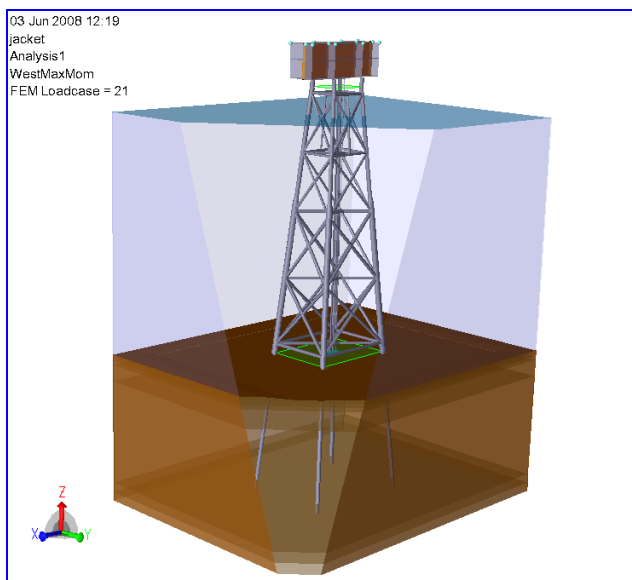
Pictures can be exported to a number of formats. From *File/Save Graphics as* you can export a picture of what you see in the graphical part of the user interface to the formats as shown to the right.



You can also increase the resolution of the pictures by increasing the width and the height of the pixels. In the example below the width is multiplied with a factor of three. Since the aspect ratio is locked the pixel number for height is automatically adjusted.

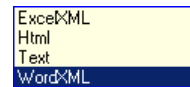


The respective pictures generated are shown below. As can be seen, the picture to the right has a better resolution since number of pixels has been increased.

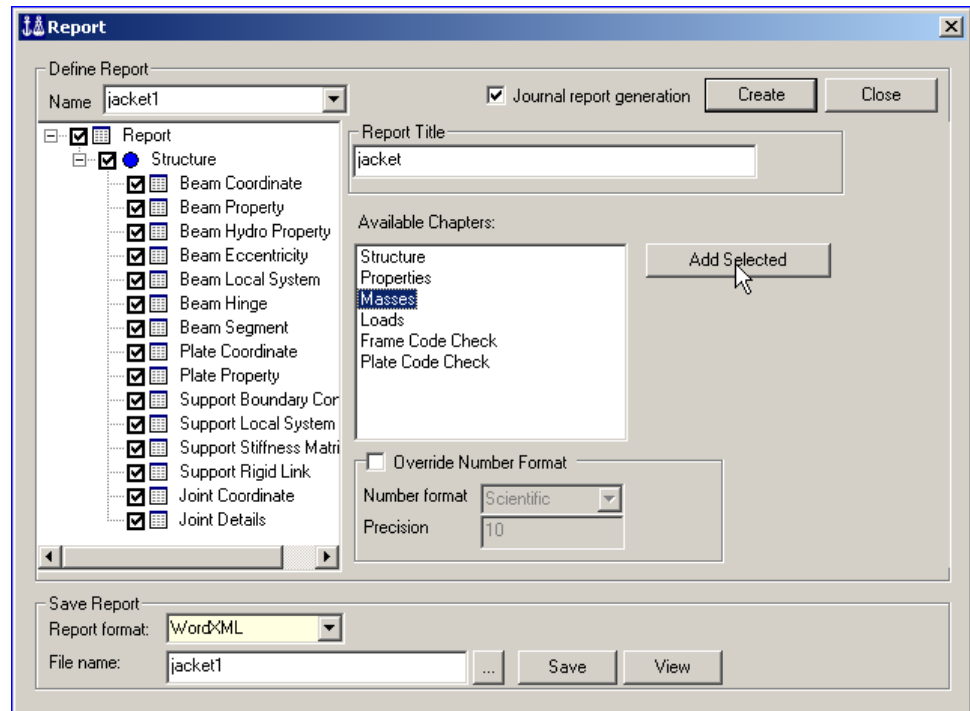


Some of the picture generation can be automated by editing a command file. Please see Chapter 5 of the User Manual Volume I on how to do this.

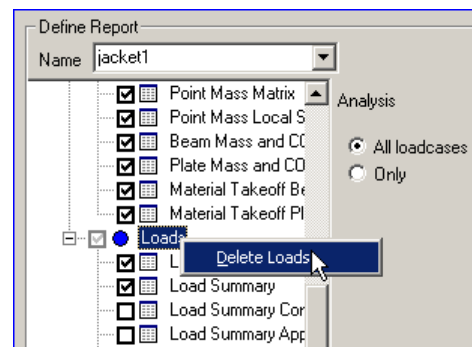
Reports can be made from **File/Save Report**. To make a report you add main Chapters and tick off for Sections within each Chapter. The report generator is scripted which means that you can recreate the same report after e.g. a model update or new analysis. The report can be exported to Word, Excel, html or a text file.



In the example below the report will contain information about the structure and the masses. The Chapter “Structure” is already selected, while the Chapter “Masses” is being added.



In case you want to remove any Chapters from a report you can select the Chapter, **RMB** and **Delete Chapter Name**.



Notice that you need Microsoft Office version 2003 or later to view the report in Excel or Word.

11. APPENDIX A: REFERENCES

Under construction.

12. APPENDIX B: MATERIAL LIBRARY

This appendix describes the materials as implemented in the XML file *Material_library.xml*. The XML file is part of standard GeniE installation. Notice that you can make your own material library the same way by a) editing an xml-file directly or b) specify all materials in GeniE and export these as Concept XML file. If you store the XML file in the c:\Program Files\DNVS\GeniE\Libraries you have direct access to it when importing a XML concept file.

The materials are listed in the API and Eurocode standards. However, this implementation is not a complete library according to the standards. The main purpose of this implementation is to give the users access to some predefined materials.

For all materials the density, Young modulus, Poisson ratio, thermal coefficient and damping factor are the same.

The materials have been implemented using units Pa (i.e. m, kg, N).

There are 71 materials in the library.

API Recommended Practice 2A-WSD									
Group	Class	Specification and Grade	Yield Strength [Pa]	Density [Kg/m ³]	Young Modulus [Pa]	Poisson Ratio	Thermal Coefficient [delC ⁻¹]	Damping Factor [N*s/m]	Tensile Strength [Pa]
I	C	ASTM_A36 (to 2 in. thick)	250	7850	2.1E11	0.3	1.2E-5	0.03	400
		ASTM_A131_Grade_A (to ½ in. thick)	235	7850	2.1E11	0.3	1.2E-5	0.03	400
		ASTM_A285_Grade_C (to ¾ in. thick)	205	7850	2.1E11	0.3	1.2E-5	0.03	380
	B	ASTM_A131 Grade_B_D	235	7850	2.1E11	0.3	1.2E-5	0.03	400
		ASTM_A516 Grade_65	240	7850	2.1E11	0.3	1.2E-5	0.03	450
		ASTM_A573 Grade_65	240	7850	2.1E11	0.3	1.2E-5	0.03	450
		ASTM_A709 Grade_36T2	250	7850	2.1E11	0.3	1.2E-5	0.03	400
	A	ASTM A131 Grade_CS_E	235	7850	2.1E11	0.3	1.2E-5	0.03	400
II	C	ASTM A572 Grade_42 (to 2 in. thick)	290	7850	2.1E11	0.3	1.2E-5	0.03	415
		ASTM A572 Grade_50 (to 2 in. thick; S91 required over ½ in.)	345	7850	2.1E11	0.3	1.2E-5	0.03	450
		API_Spec_2MT2_ Class_C	345	7850	2.1E11	0.3	1.2E-5	0.03	450
		ASTM_A992	345	7850	2.1E11	0.3	1.2E-5	0.03	450
	B	API_Spec_2MT1	345	7850	2.1E11	0.3	1.2E-5	0.03	483
		API_Spec_2MT2 Class_B	345	7850	2.1E11	0.3	1.2E-5	0.03	450
		ASTM_A709 Grade_50T2_50T3	345	7850	2.1E11	0.3	1.2E-5	0.03	450

		ASTM_A131 Grade_AH32	315	7850	2.1E11	0.3	1.2E-5	0.03	470
		ASTM_A131 Grade_AH36	350	7850	2.1E11	0.3	1.2E-5	0.03	490
A		API_Spec_2H Grade_42	290	7850	2.1E11	0.3	1.2E-5	0.03	430
		API_Spec_2H Grade50 (to 2½ in. thick)	345	7850	2.1E11	0.3	1.2E-5	0.03	483
		API_Spec_2W Grade42(to 1 in. thick)	290	7850	2.1E11	0.3	1.2E-5	0.03	427
		API_Spec_2W Grade42(to 1 in. thick)	290	7850	2.1E11	0.3	1.2E-5	0.03	427
		API_Spec_2W Grade50(to 1 in. thick)	345	7850	2.1E11	0.3	1.2E-5	0.03	448
		API_Spec_2W Grade50T(to 1 in. thick)	345	7850	2.1E11	0.3	1.2E-5	0.03	483
		API_Spec_2Y Grade42(to 1 in. thick)	290	7850	2.1E11	0.3	1.2E-5	0.03	427
		API_Spec_2Y Grade50(to 1 in. thick)	345	7850	2.1E11	0.3	1.2E-5	0.03	448
		API_Spec_2Y Grade50T(to 1 in. thick)	345	7850	2.1E11	0.3	1.2E-5	0.03	483
		API_Spec_2MT2 Class_A	345	7850	2.1E11	0.3	1.2E-5	0.03	450
		ASTM_A131 Grade_DH32_EH32	315	7850	2.1E11	0.3	1.2E-5	0.03	470
		ASTM_A131 Grade_DH36_EH36	350	7850	2.1E11	0.3	1.2E-5	0.03	490
		ASTM_A537 Class_I (to 2½ in. thick)	345	7850	2.1E11	0.3	1.2E-5	0.03	485
		ASTM_A633 Grade_A	290	7850	2.1E11	0.3	1.2E-5	0.03	435
		ASTM_A633 Grade_C_D	345	7850	2.1E11	0.3	1.2E-5	0.03	485
		ASTM_A678 Grade_A	345	7850	2.1E11	0.3	1.2E-5	0.03	485
		ASTM_A913_Grade_5 0 (with CVN @-20°C)	345	7850	2.1E11	0.3	1.2E-5	0.03	450

III	A	ASTM_A537 Class_II (to 2½ in. thick)	415	7850	2.1E11	0.3	1.2E-5	0.03	550
		ASTM_A678 Grade_B	415	7850	2.1E11	0.3	1.2E-5	0.03	550
		API_Spec 2W Grade_60(to 1 in. thick)	414	7850	2.1E11	0.3	1.2E-5	0.03	517
		API_Spec 2Y Grade_60(to 1 in. thick)	414	7850	2.1E11	0.3	1.2E-5	0.03	517
		ASTM_A710_Grade_ A Class_3 (through 2in.)	515	7850	2.1E11	0.3	1.2E-5	0.03	585

EUROCODE								
Standard	Grade	Yield Strength [Pa]	Density [Kg/m ³]	Young Modulus [Pa]	Poisson Ratio	Thermal Coefficient [delC ⁻¹]	Damping Factor [N*s/m]	Tensile Strength [Pa]
EN 10025-2	S235	235	7850	2.1E11	0.3	1.2E-5	0.03	360
	S275	275	7850	2.1E11	0.3	1.2E-5	0.03	430
	S355	355	7850	2.1E11	0.3	1.2E-5	0.03	510
	S450	440	7850	2.1E11	0.3	1.2E-5	0.03	550
EN 10025-3	S275_N_NL	275	7850	2.1E11	0.3	1.2E-5	0.03	390
	S355_N_NL	355	7850	2.1E11	0.3	1.2E-5	0.03	490
	S420_N_NL	420	7850	2.1E11	0.3	1.2E-5	0.03	520
	S460_N_NL	460	7850	2.1E11	0.3	1.2E-5	0.03	540
EN 10025-4	S275_M_ML	275	7850	2.1E11	0.3	1.2E-5	0.03	370
	S355_M_ML	355	7850	2.1E11	0.3	1.2E-5	0.03	470
	S420_M_ML	420	7850	2.1E11	0.3	1.2E-5	0.03	520
	S460_M_ML	460	7850	2.1E11	0.3	1.2E-5	0.03	540
EN 10025-5	S235_W	235	7850	2.1E11	0.3	1.2E-5	0.03	360
	S355_W	355	7850	2.1E11	0.3	1.2E-5	0.03	510
EN 10025-6	S460_Q_QL_QL1	460	7850	2.1E11	0.3	1.2E-5	0.03	570
EN 10210-1	S235_H	235	7850	2.1E11	0.3	1.2E-5	0.03	360
	S275_H	275	7850	2.1E11	0.3	1.2E-5	0.03	430
	S355_H	355	7850	2.1E11	0.3	1.2E-5	0.03	510

	S275_NH_NLH	275	7850	2.1E11	0.3	1.2E-5	0.03	390
	S355_NH_NLH	355	7850	2.1E11	0.3	1.2E-5	0.03	490
	S420_NH_NLH	420	7850	2.1E11	0.3	1.2E-5	0.03	540
	S460_NH_NLH	460	7850	2.1E11	0.3	1.2E-5	0.03	560
EN 10219-1	S235H	235	7850	2.1E11	0.3	1.2E-5	0.03	360
	S275H	275	7850	2.1E11	0.3	1.2E-5	0.03	430
	S355H	355	7850	2.1E11	0.3	1.2E-5	0.03	510
	S275_NH_NLH	275	7850	2.1E11	0.3	1.2E-5	0.03	370
	S355_NH_NLH	355	7850	2.1E11	0.3	1.2E-5	0.03	470
	S460_NH_NLH	460	7850	2.1E11	0.3	1.2E-5	0.03	550
	S275_MH_MLH	275	7850	2.1E11	0.3	1.2E-5	0.03	360
	S355_MH_MLH	355	7850	2.1E11	0.3	1.2E-5	0.03	470
	S420_MH_MLH	420	7850	2.1E11	0.3	1.2E-5	0.03	500
	S460_MH_MLH	460	7850	2.1E11	0.3	1.2E-5	0.03	530